

L5: ALTIUM SCHEMATICS I

ESE516: IoT Edge Computing

Wednesday February 6, 2019

Eduardo Garcia - edgarc@seas.upenn.edu

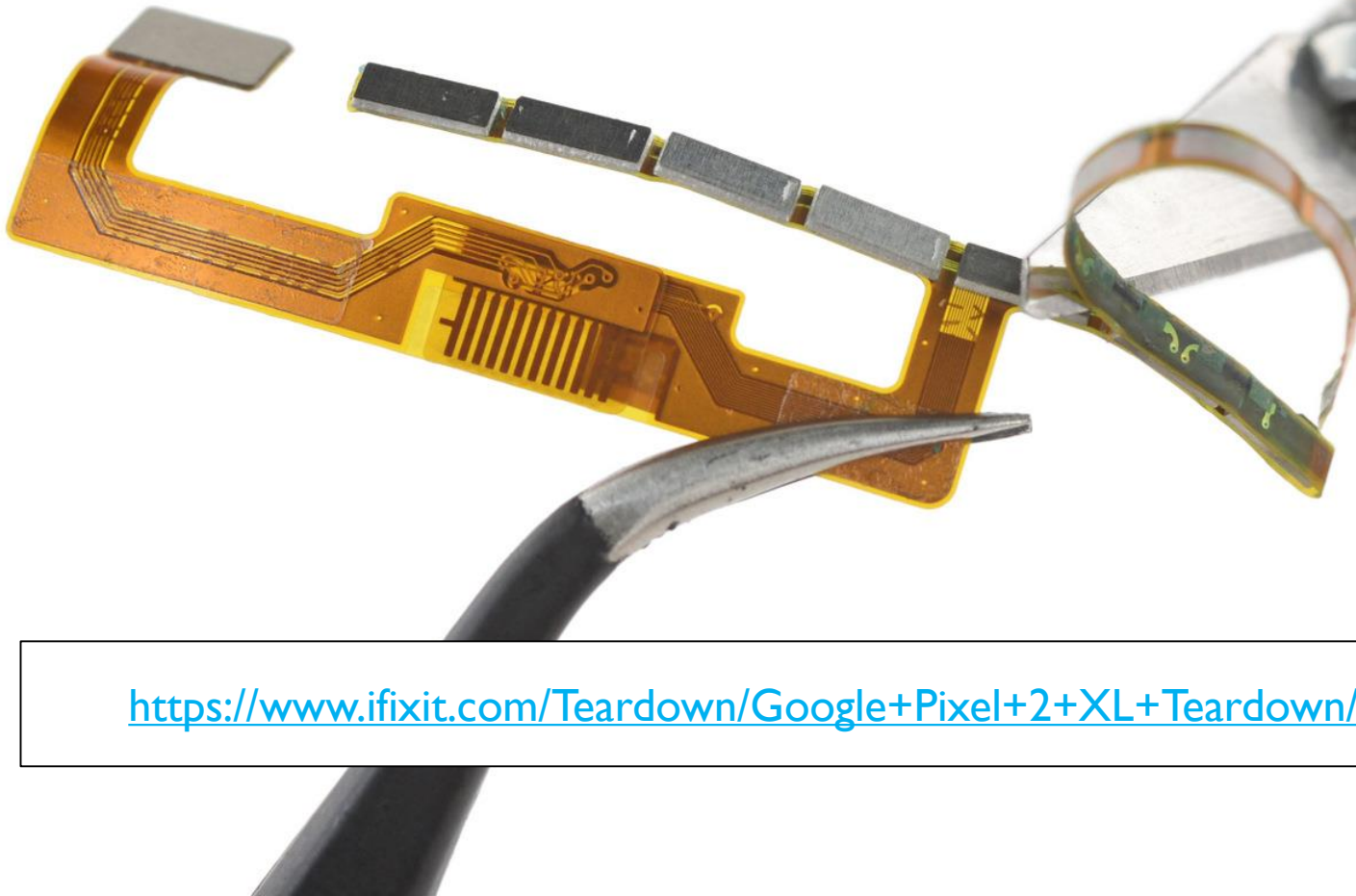
ADMINISTRATIVE

A2

- A2 is posted – please check Piazza

TODAY'S LECTURE

GOOGLE PIXEL 2 XL



<https://www.ifixit.com/Teardown/Google+Pixel+2+XL+Teardown/98093>

LECTURE GOALS

Altium
Designer®

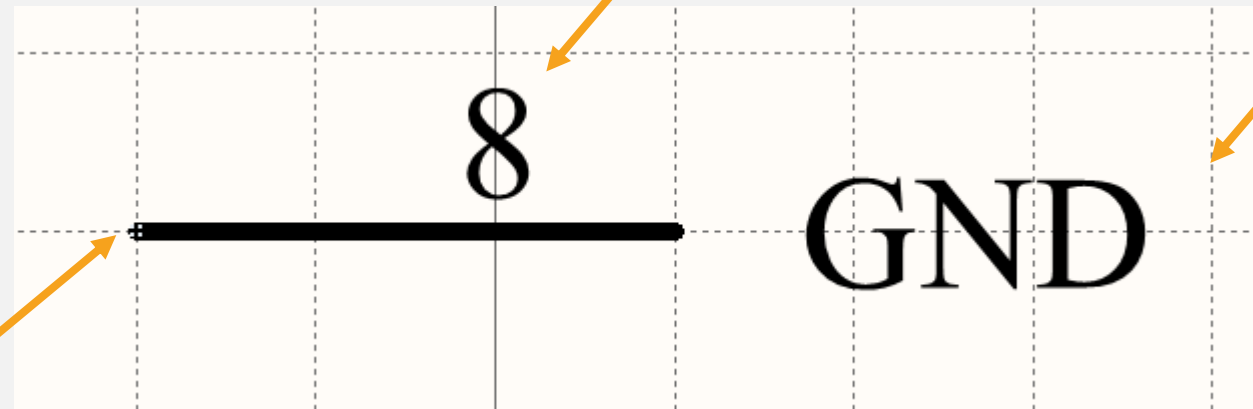
- Solve Altium Component Questions
- Good practices and Tips on Component Creation
- Introduction to Atmel Schematic Editor

COMPONENT BEST PRACTICES

COMPONENT DRAWING - PIN

Designator –
Unique Pin Name
(Used to assign a
pin to a copper
pad(s) on the
PCB Footprint)

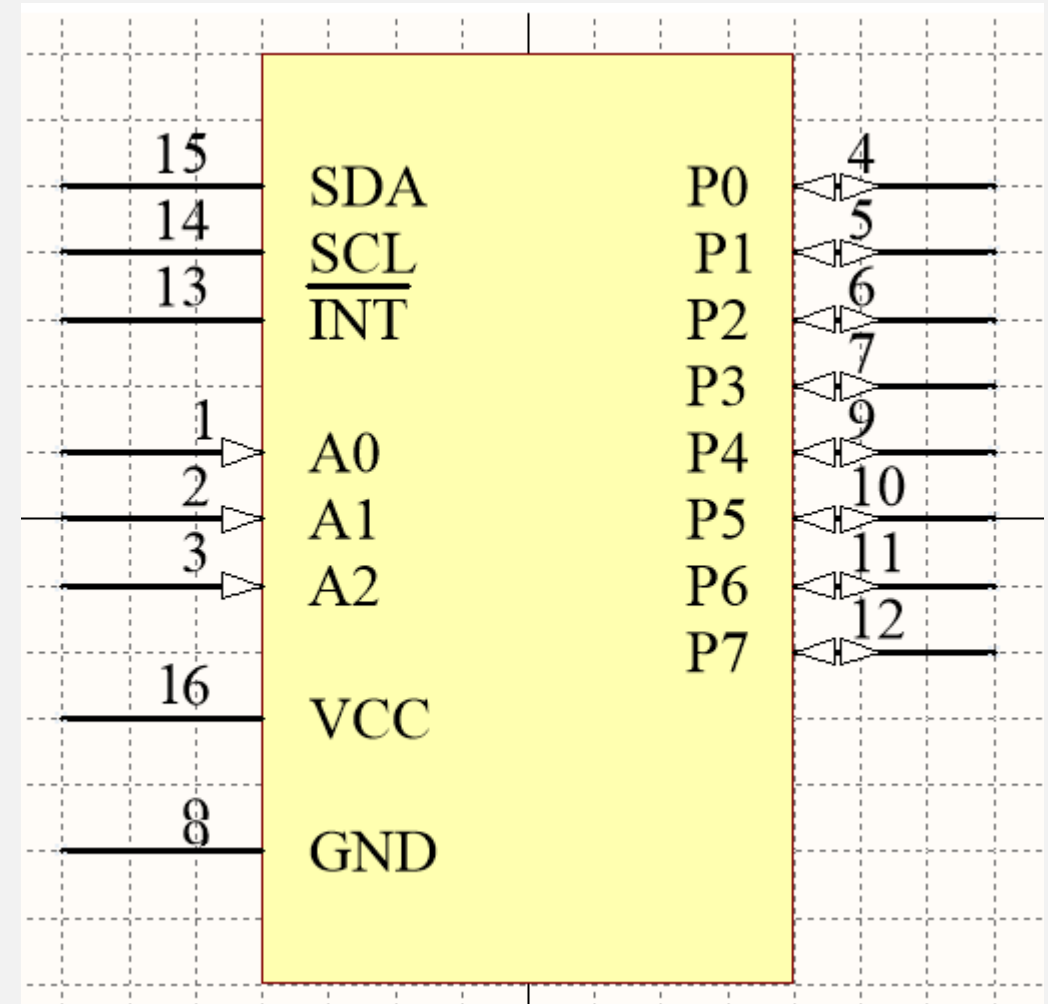
Name –
descriptive name
of Pin function



Logical connection (where
Altium decided the pin
connects to wires)

COMPONENT DRAWING

- Good practices:
 - Pins align with intersection of Grid (Grid of 50 mil)
 - Draw the component symbol close to the sheet origin (the center of the sheet).
 - Descriptive Pin Names. If more than one function, write all of them
- Do logical layout

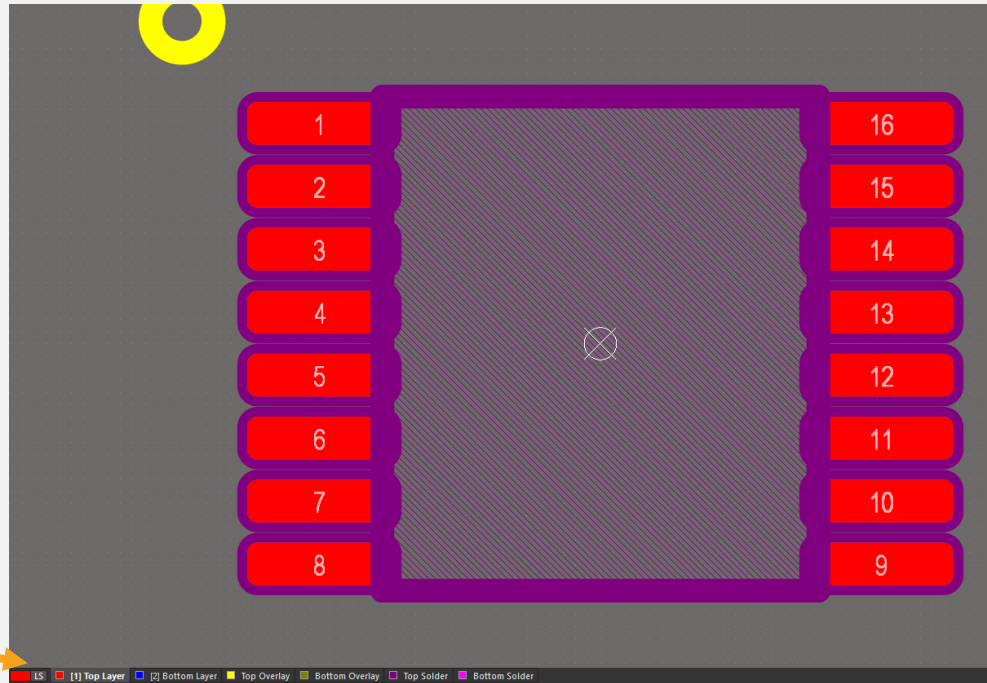


FOOTPRINT TIPS

Tip: Shift + S cycles through the following:

- Show current layer, shadow others
- Show only current layer
- Normal view (all layers)

Click on layer tab to bring them to the front of the drawing (so you can see better!)

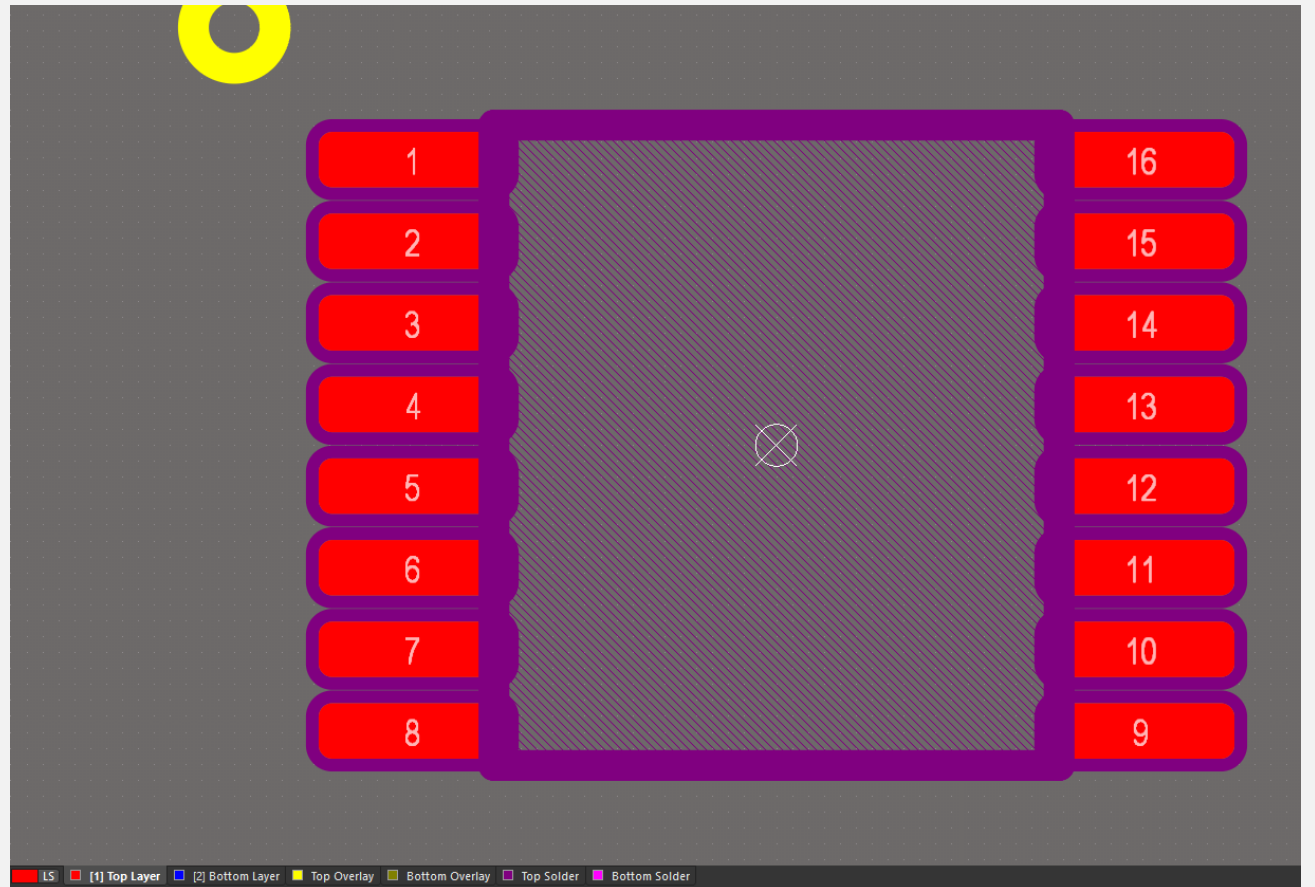


Tip: Altium uses “mm” or “mil” as its units of measurements. Press ‘Q’ to cycle through them.

The coordinates on the bottom left corner of the screen will show you what unit of measurement is currently used

FOOTPRINT TIPS

- Good practices:
 - Make sure origin is on the center of the component
 - If you drew the component off center, you can do “Edit -> Set Reference -> Center” to place it on the center of what you drew.
 - Always add a silkscreen indicator for the pin 1



QUESTIONS ON COMPONENT FABRICATION?

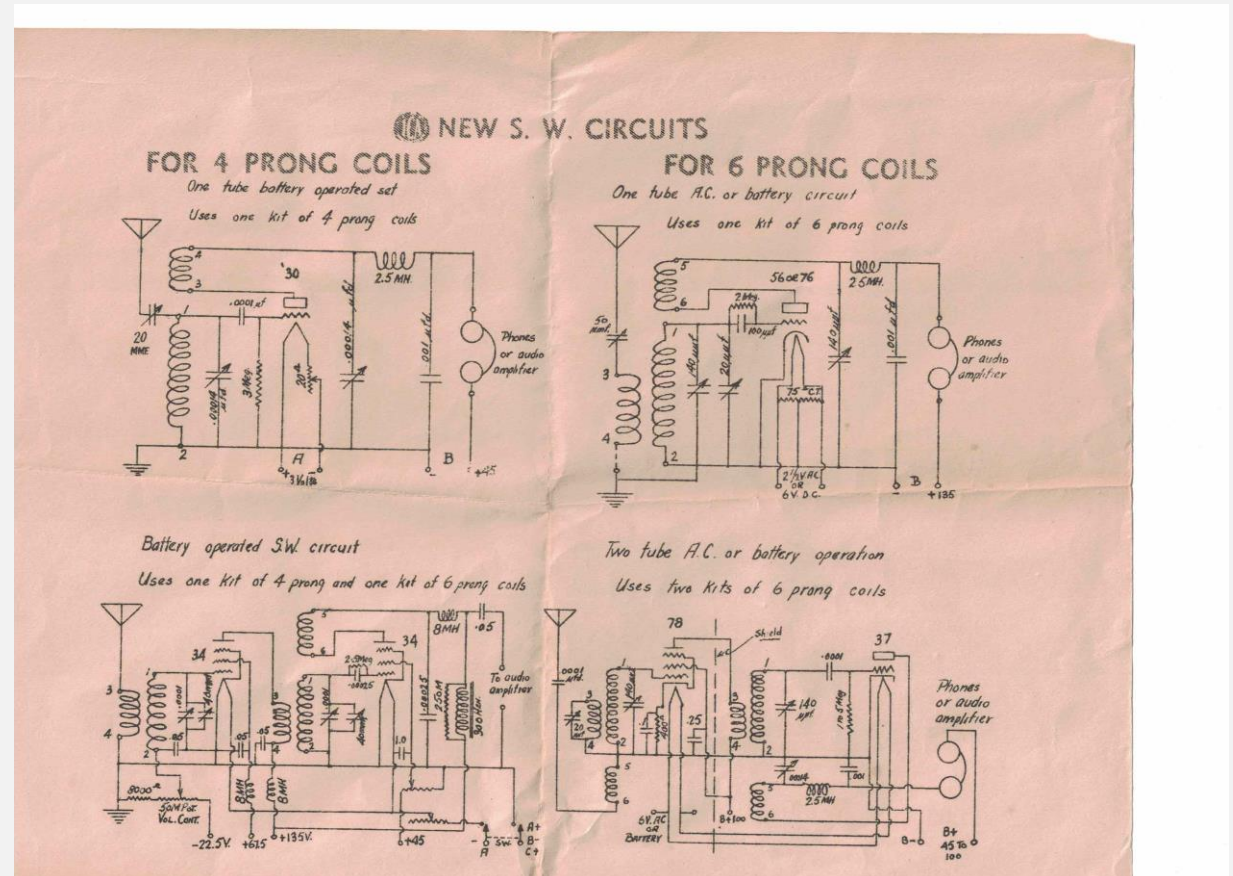
TIPS GOING FORWARD

- You can find component on the Altium Content Vault (we will see this in this lecture). First search for your component there to see if it is already there – that will save you a lot of work!
- Use SnapEDA or the manufacturer's website to (possibly) find components made for you
- Always double-check that everything you downloaded is correct!

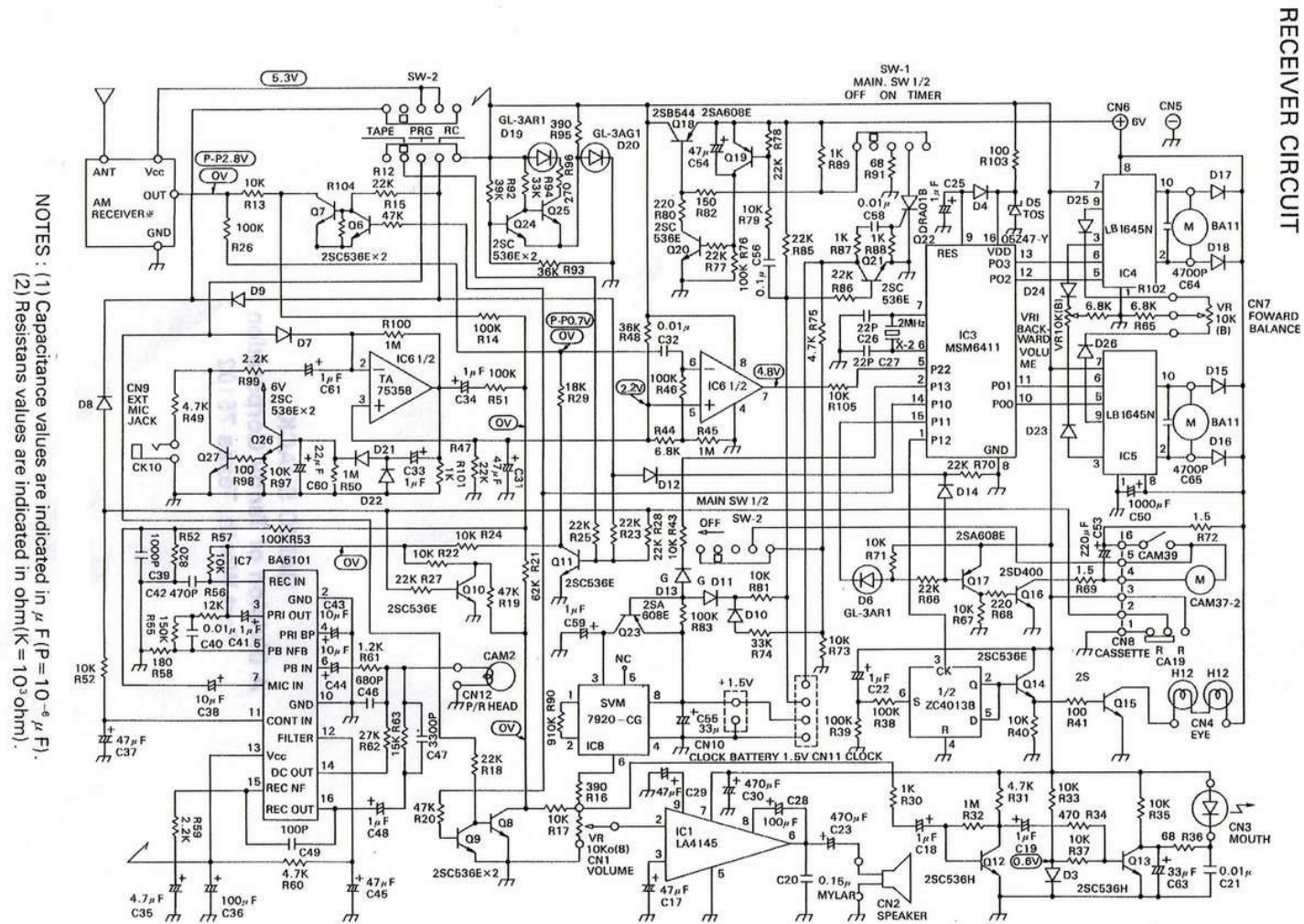
SCHEMATICS – GENERAL INTRODUCTION

SCHEMATICS OVER TIME

- Electrical schematics and how to draw them change over time as tools improve
- On the Old Times, schematics were usually drawn with all the connections on one paper
- Today, EDA facilitates drawing schematics modularly, which may facilitate reading, if done correctly

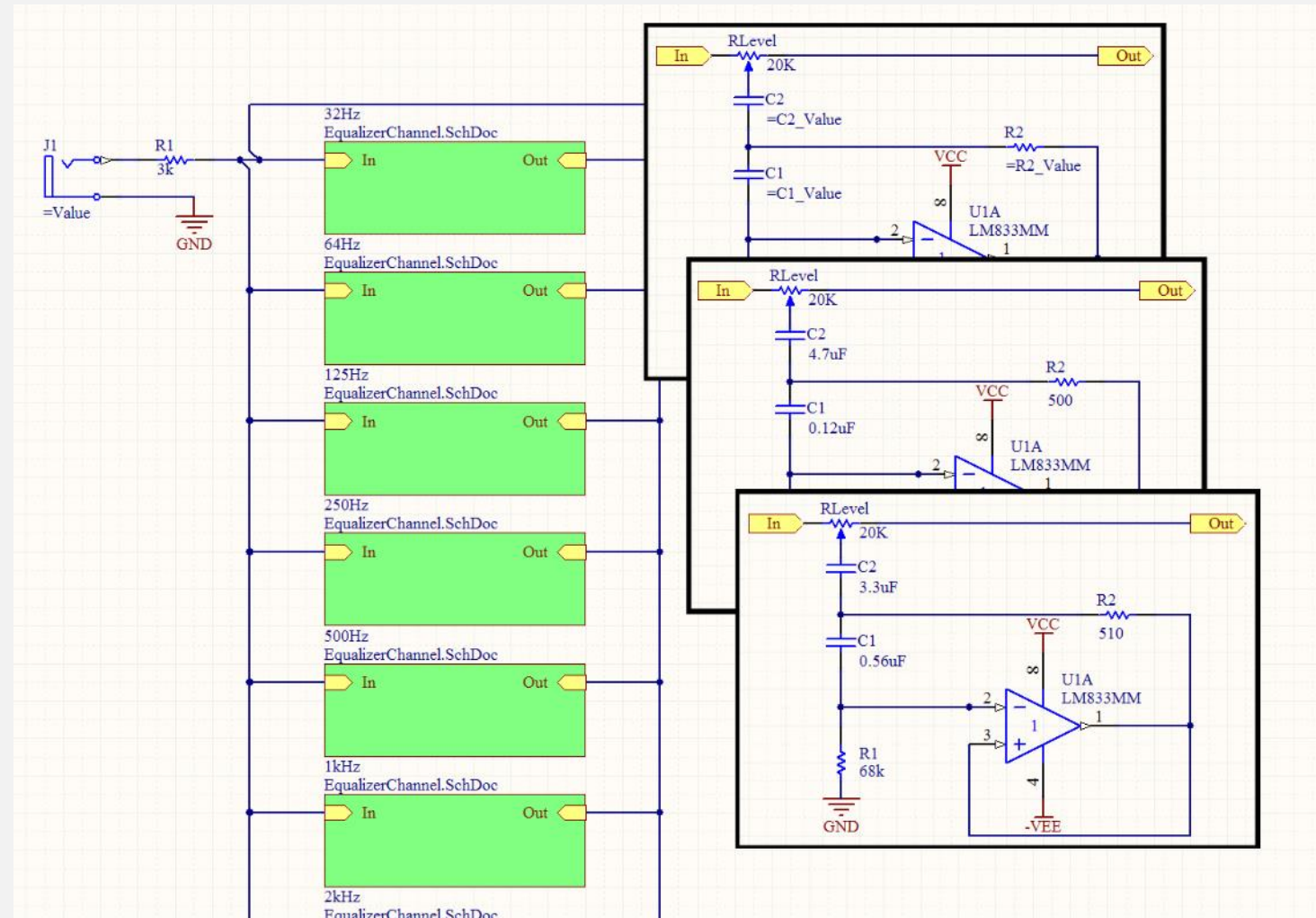


SCHEMATICS- ALL IN A SHEET



SCHEMATICS - MODULAR

- Multi-Sheet design: You can make a “black box” out of one sheet and use it as a module in another sheet
- This reduces complexity and allows modularization
- This will be our preferred method of designing our boards.

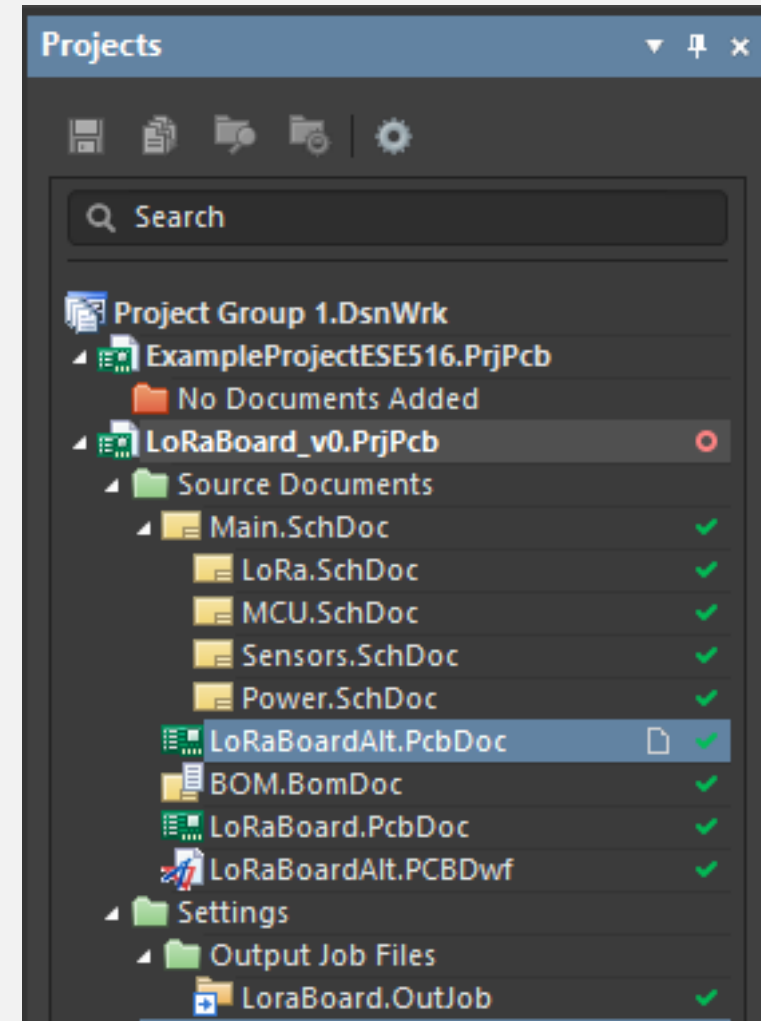


ALTium PROJECTS

SCHEMATICS - MODULAR

- You can create an Altium Project for your project.
- A project allows you to associate schematics, boards, and other useful files to your design. It is a similar idea to having an Atmel Studio Project.
- Projects, once opened, will appear on your “Project” Tab and will list all the files associated with your project

Project Tree for the
Project “LoRaBoard_v0”



ALTium SCHEMATIC TUTORIAL I – MAKING A PROJECT

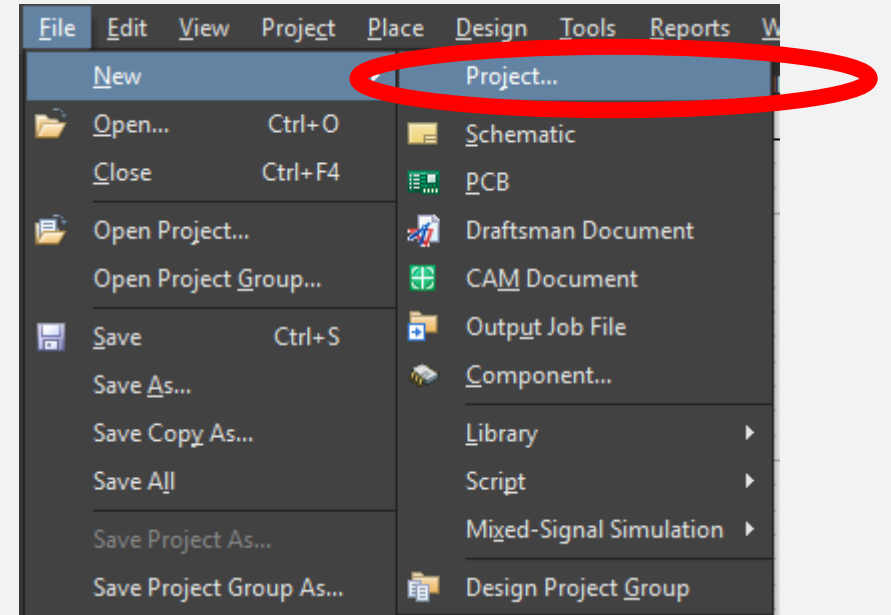
SCHEMATIC LIBRARY

STARTING OUT

- In this tutorial we will do the following:
 - Learn how to make a project
 - Learn how to add an schematic to our project
 - Learn how to add components from the Altium Content Vault
 - Learn how to add components from a Schematic Library
 - Learn how to wire a circuit
 - Learn how to use an schematic to make a sub circuit and use it as a module in another schematic

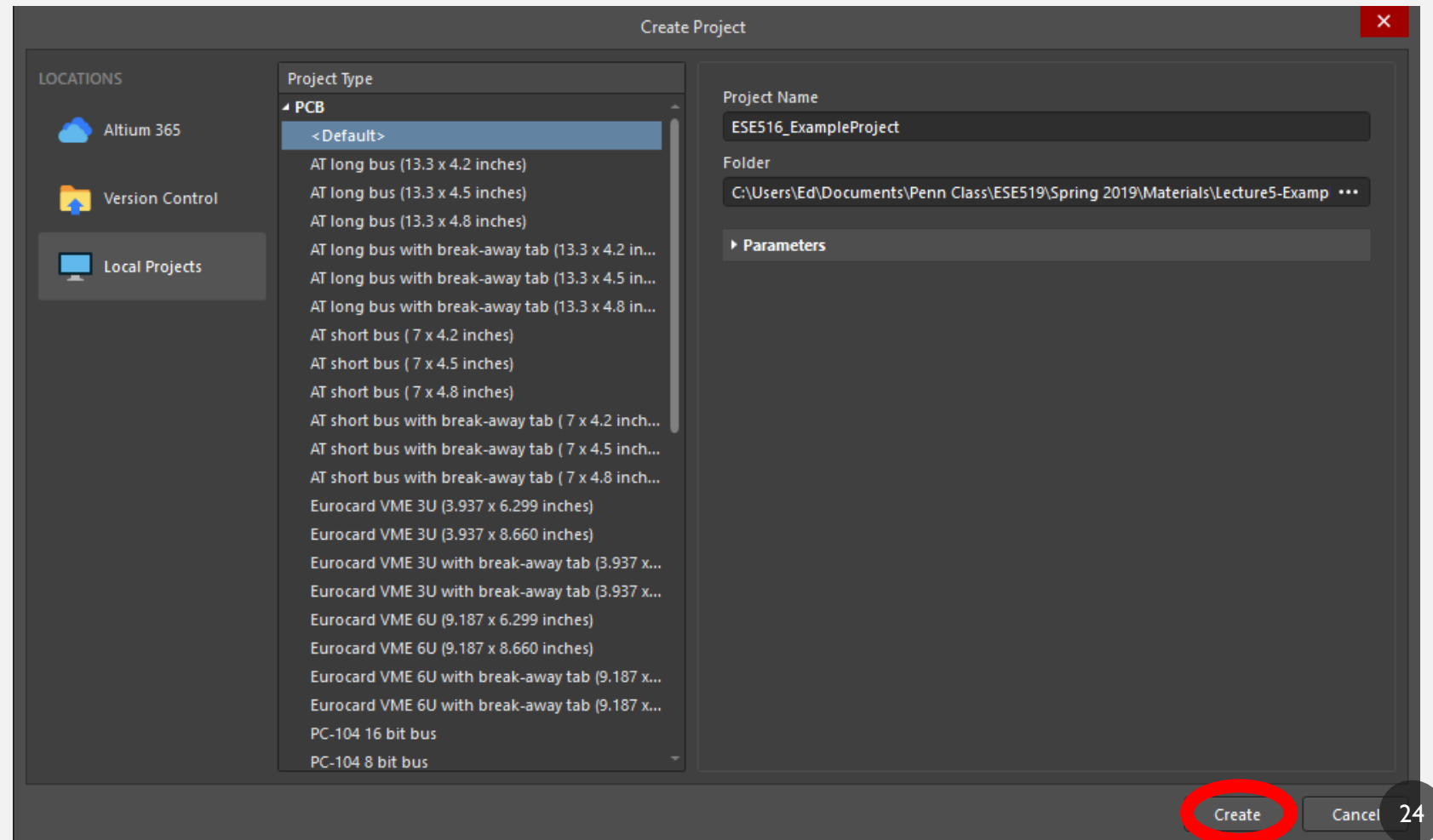
MAKING A NEW PROJECT

- Open Altium and get a license.
- Go to “File” -> “New” -> “Project...”

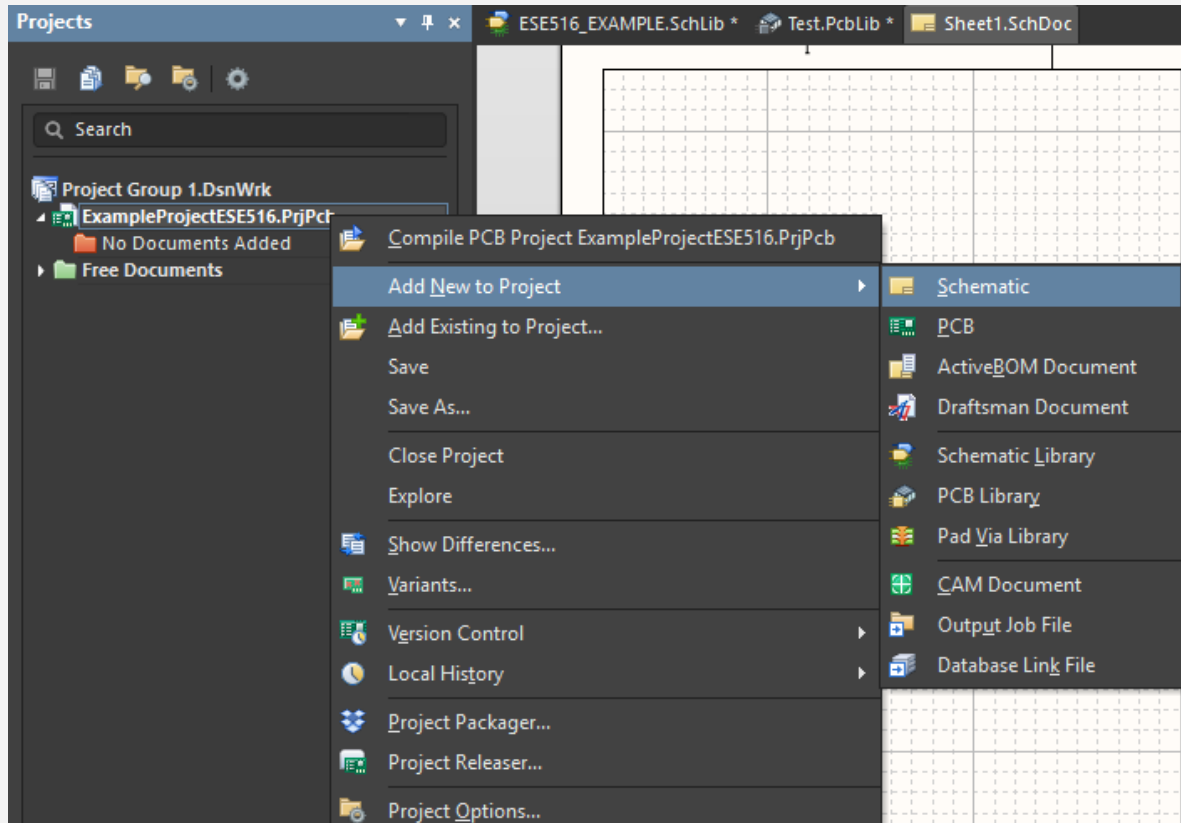


MAKING A NEW PROJECT

- On Project Type, choose <Default>. The other types are rarely used! But if you are going to make a PCB in one of the listed formats in the future, you can choose it here to get a template of the PCB shape added for you.
- Please save the project locally – if you try to do it on the S: drive it might get slow.
- Hit Create

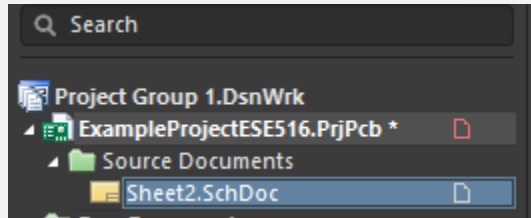


MAKING A NEW PROJECT

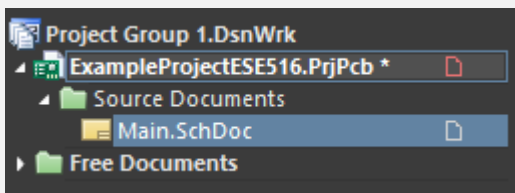
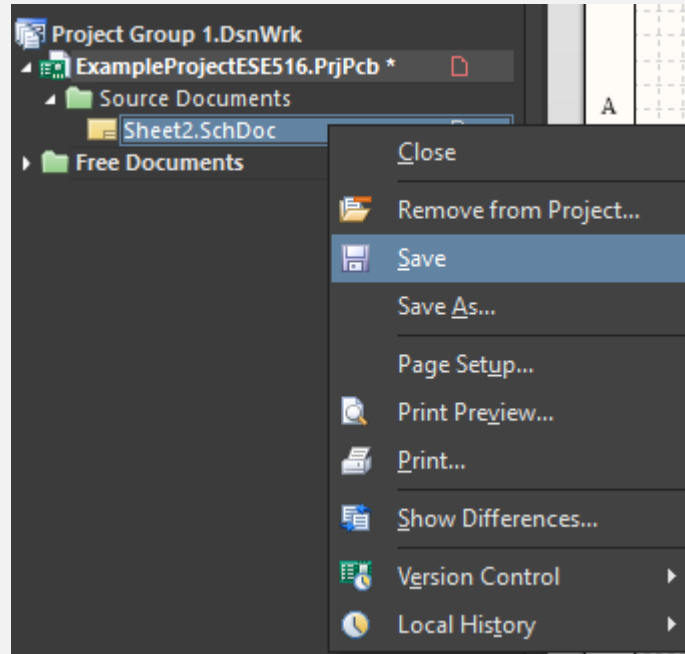


- You should have the project you just created on the “projects” tab.
- To add our first schematic file to our project, right-click the project and go to “Add New to Project -> Schematic”

MAKING A NEW PROJECT



Right click

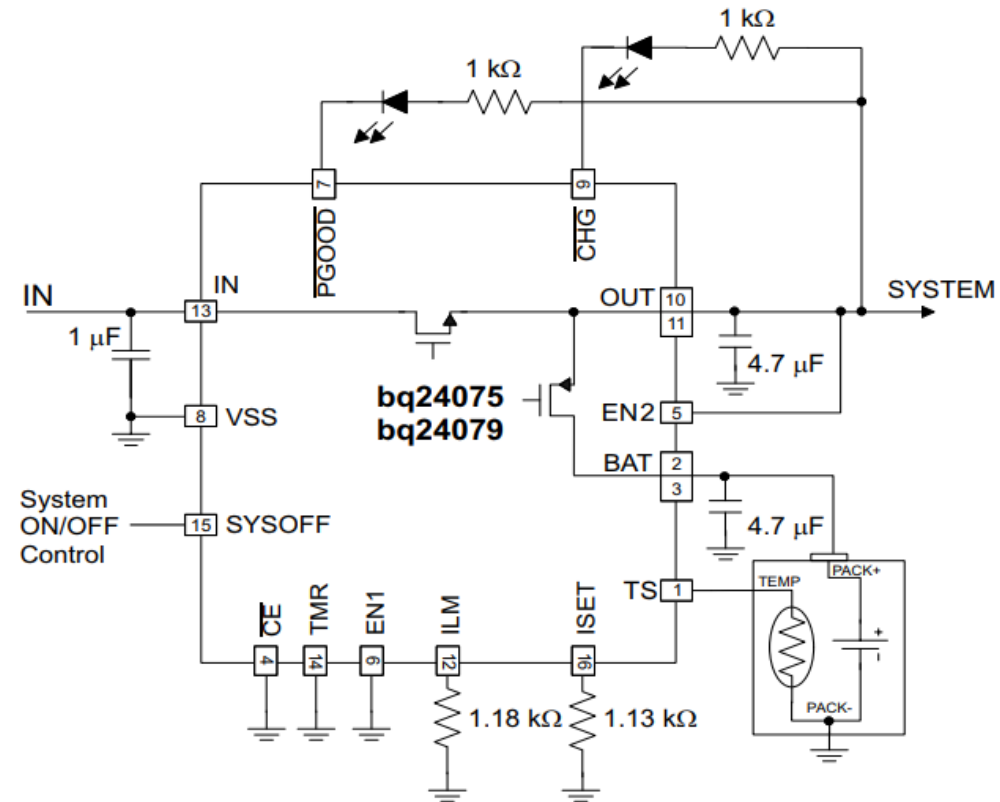


- You should see a schematic sheet added to your project (Sheet1.schdoc or a similar name). Let's save the sheet and give it a more descriptive name.
- **Tip: Usually the top sheet is called “Main”.** This indicates it is the top sheet!

YOUR FIRST SHEET

- You now have a sheet that is empty. We will use this sheet as the top level sheet, which will have the complete circuit connection.
- To start, let's try to make the application circuit of the "BQ24075". This is a LiPo charger IC that does the following:
 - Can accept a voltage input from a LiPo or an USB connection
 - Will connect the LiPo to the output if the USB is not present
 - Will use the USB voltage if the USB is present AND will disconnect the LiPo from the output and charge it.

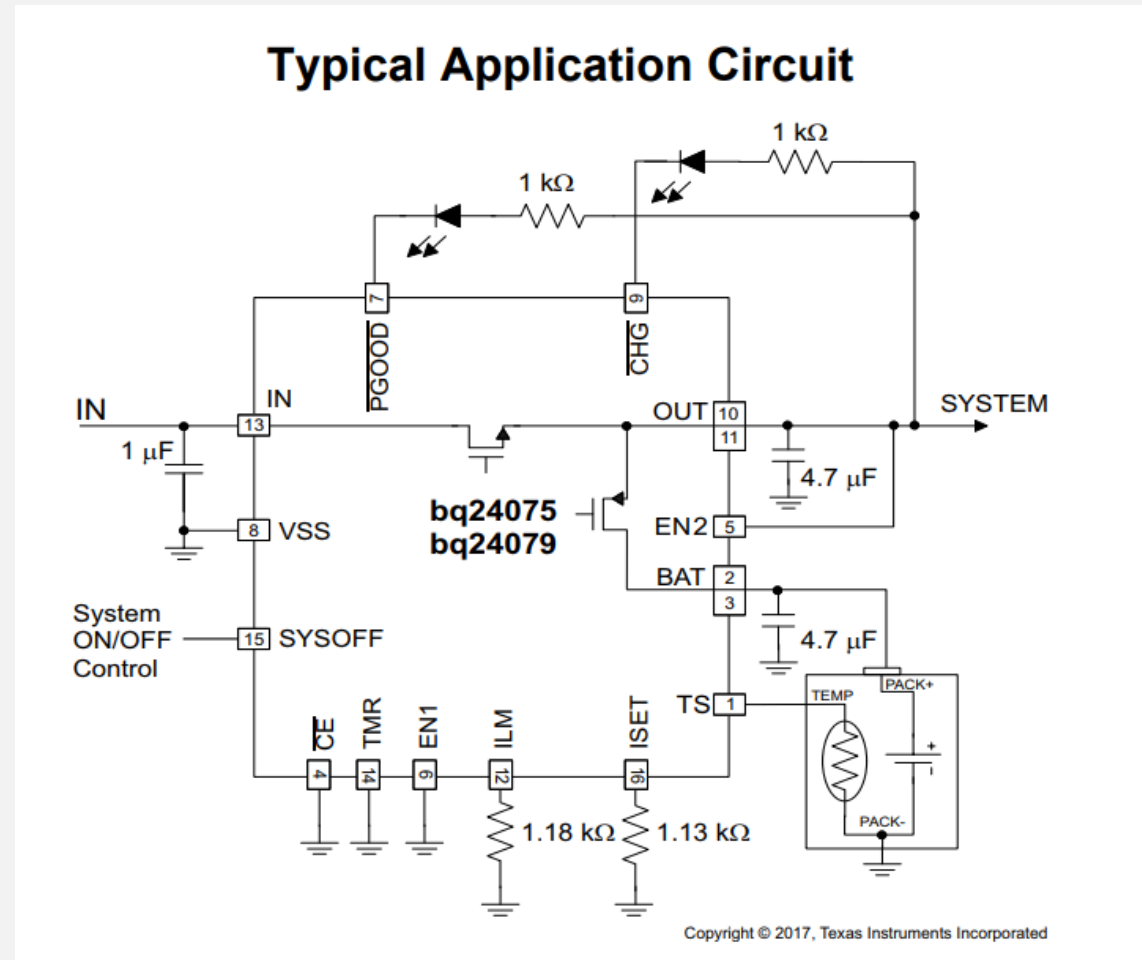
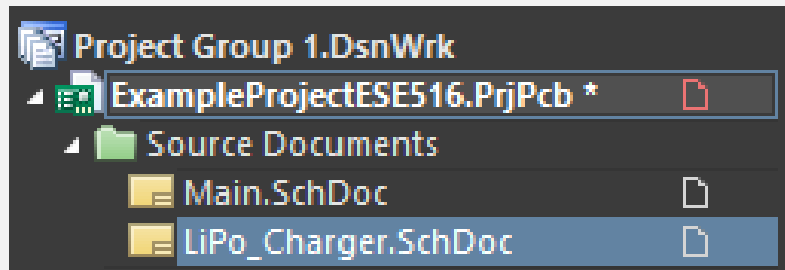
Typical Application Circuit



Copyright © 2017, Texas Instruments Incorporated

YOUR FIRST SHEET

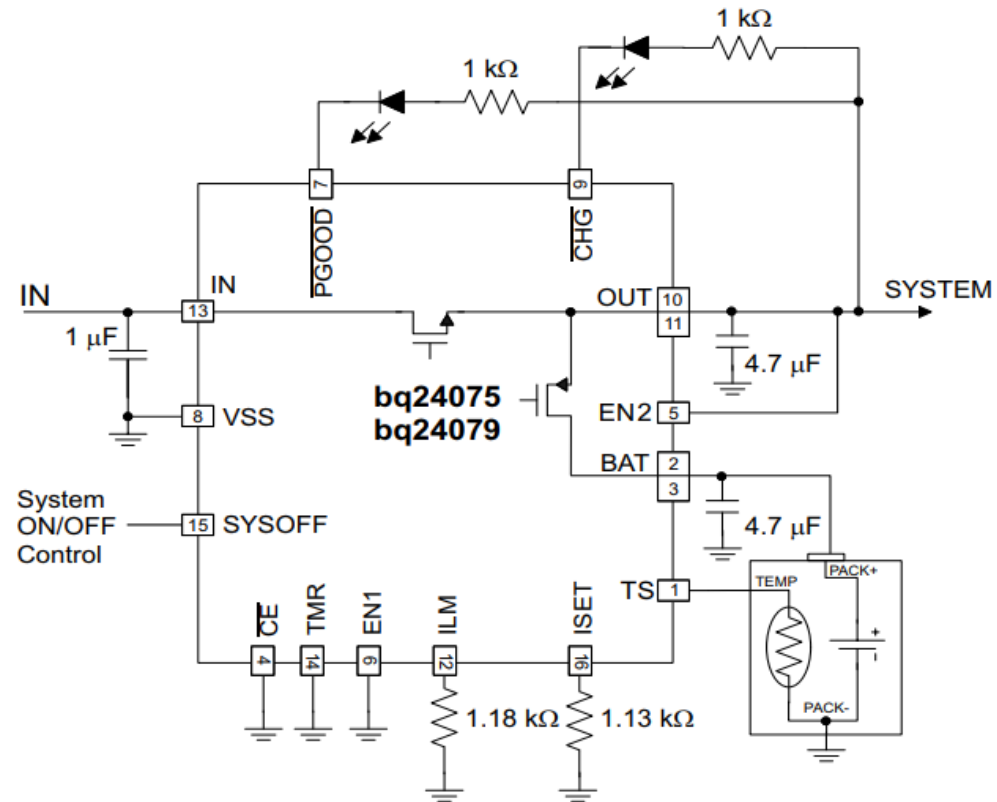
- To start, open the BQ24075 datasheet and read the first page (<http://www.ti.com/lit/ds/symlink/bq24075.pdf>)
- Make a new schematic and call it “LiPo_Charger”
- Your project Tree will look as the following:



YOUR FIRST SHEET

- You might notice – Do we have to do all those components ourselves?
- The answer is – they might be available in Altium's Content Vault!
- Altium's Content Vault is a repository where Altium provides some components that you can use on your design from different manufacturers.
- Manufacturers like TI are really good at providing Altium with their components. Other manufacturers – not so much!
- If you can't find it on the vault, you have to find it elsewhere or do it yourself...
- Altium provide footprint for very common components – Resistors, Capacitors, Inductors, Leds, Diodes, ETC.

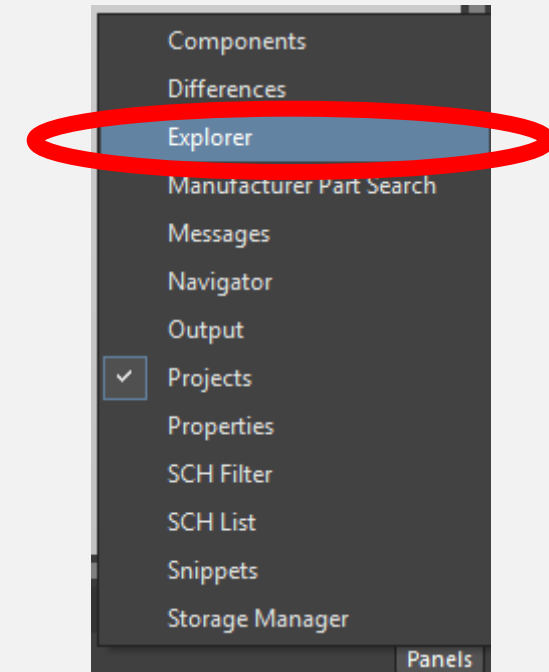
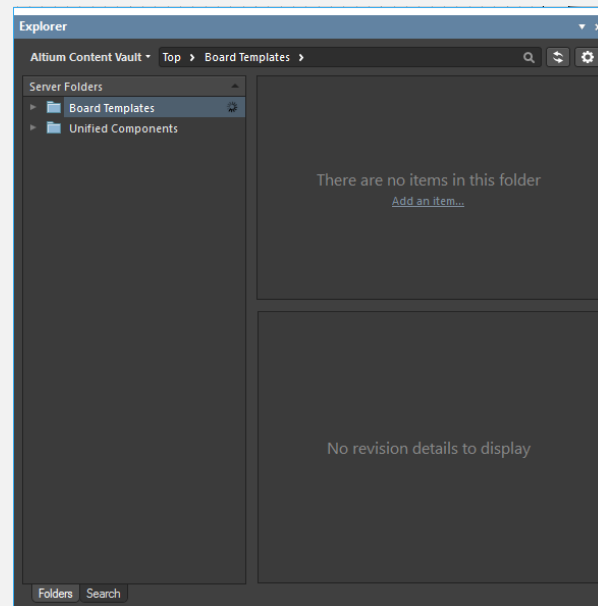
Typical Application Circuit



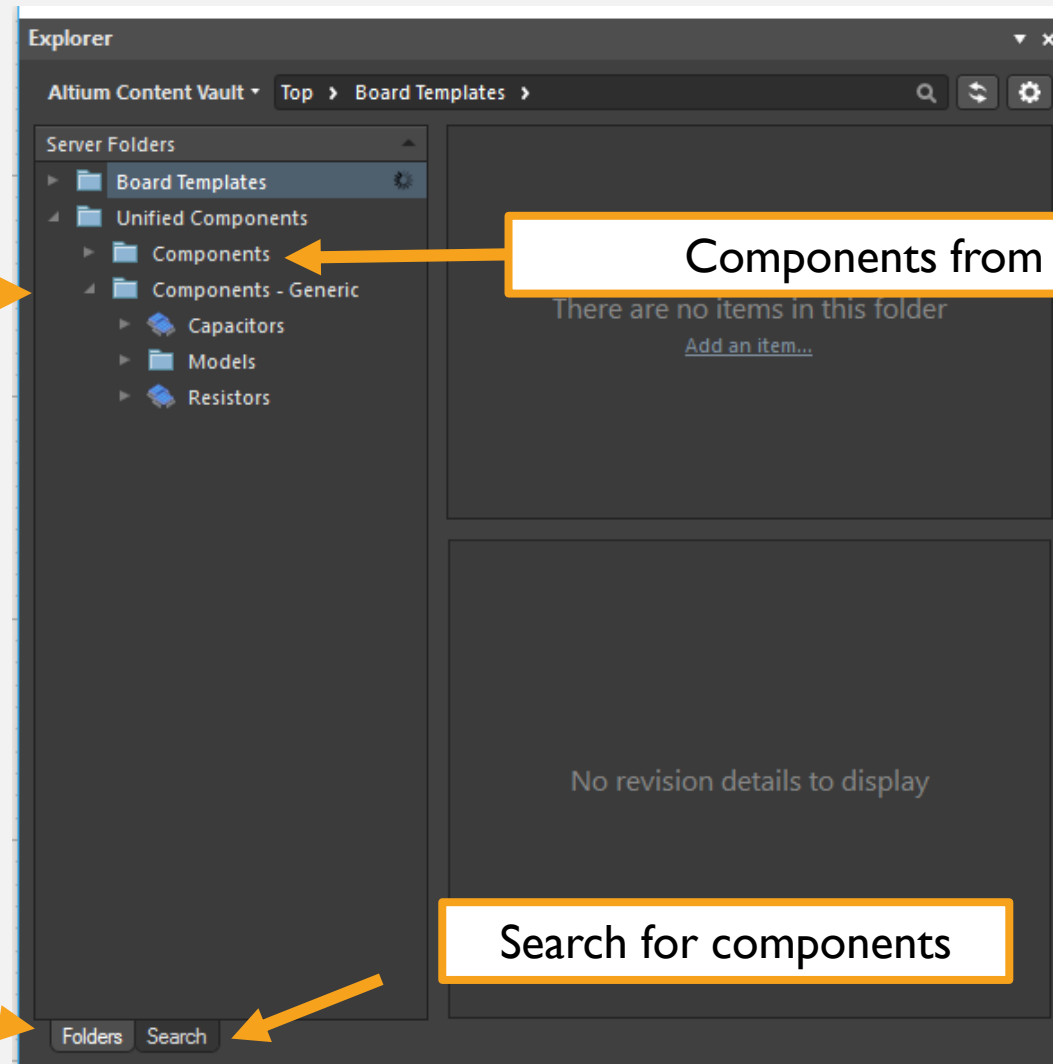
Copyright © 2017, Texas Instruments Incorporated

OPENING ALTIUM CONTENT VAULT (OR EXPLORER)

- To open the Altium Content Vault on Altium 19, Go to the “Panels” button on the bottom right, and search for “Explorer”.
- This will open the Explorer Window, which may be a free-floating window



ALTIUM CONTENT VAULT (OR EXPLORER)



Components from manufacturers

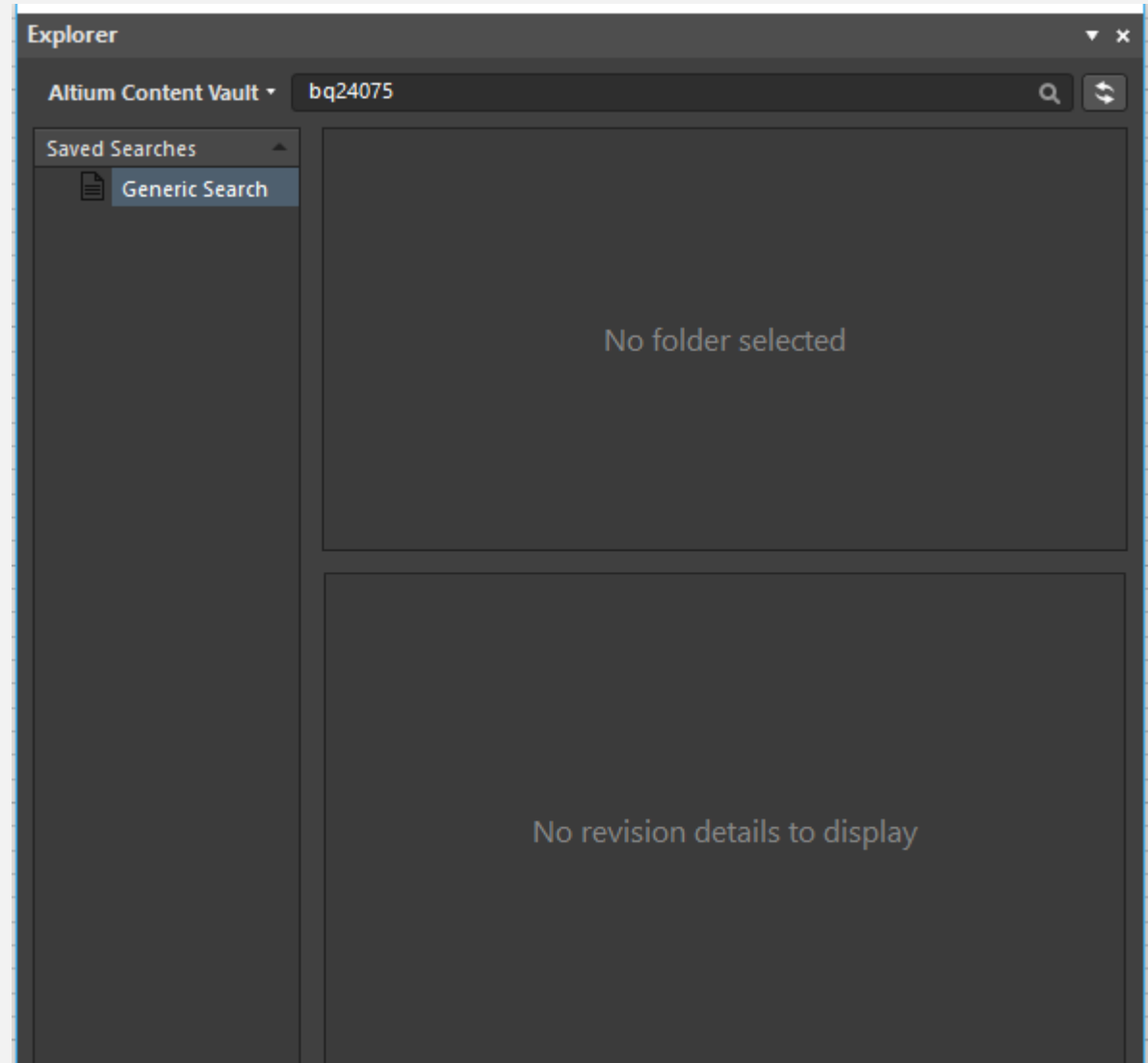
Generic components
for Capacitors and
Resistors

Explore the components

Search for components

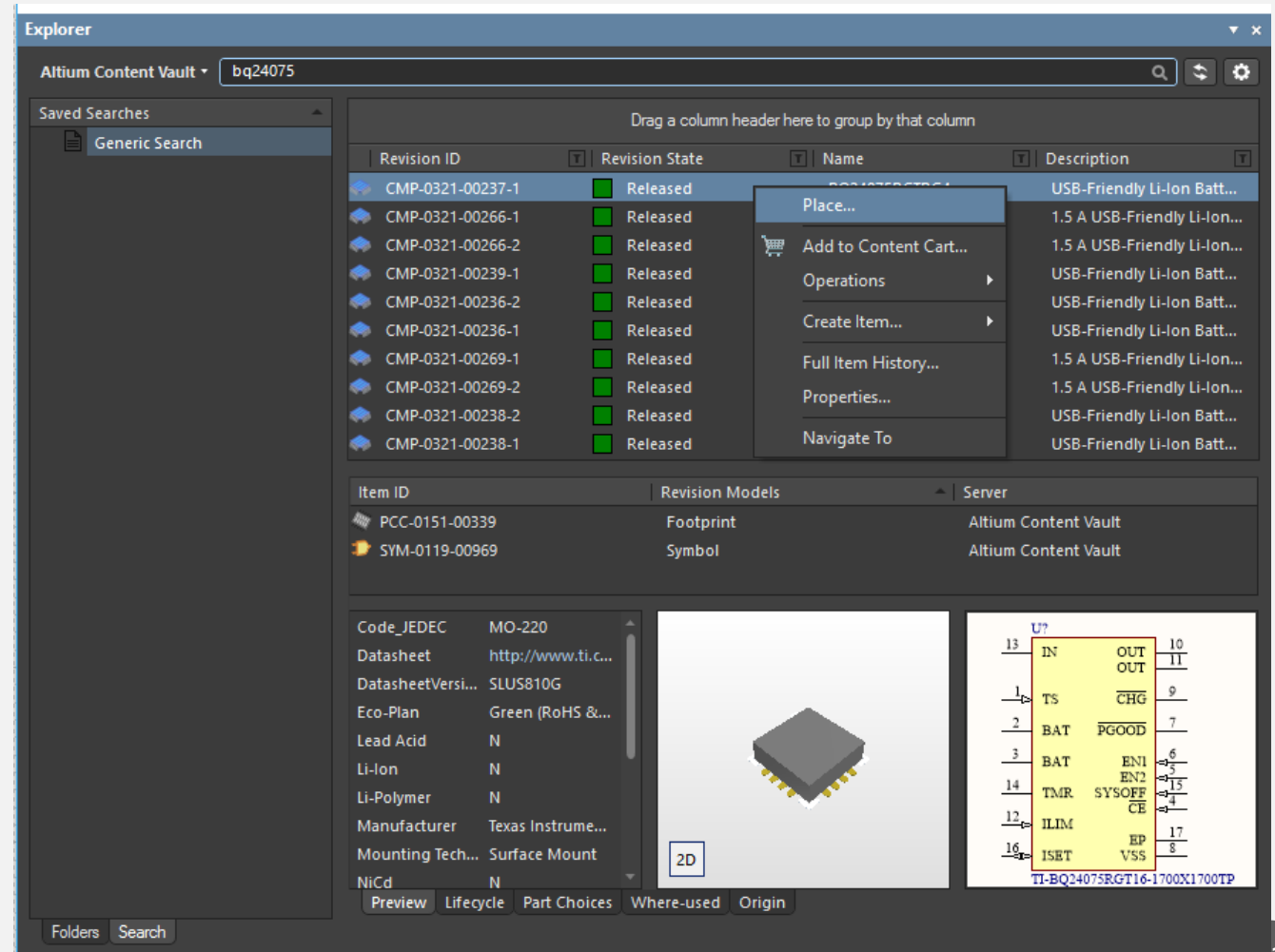
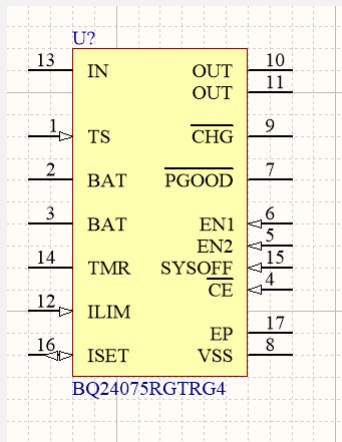
ADDING A COMPONENT FROM THE ALTIUM VAULT

- Let's check first if the Altium Content Vault has the BQ24075. For this, go to the “search” tab on the and type “”. Hit Enter to search



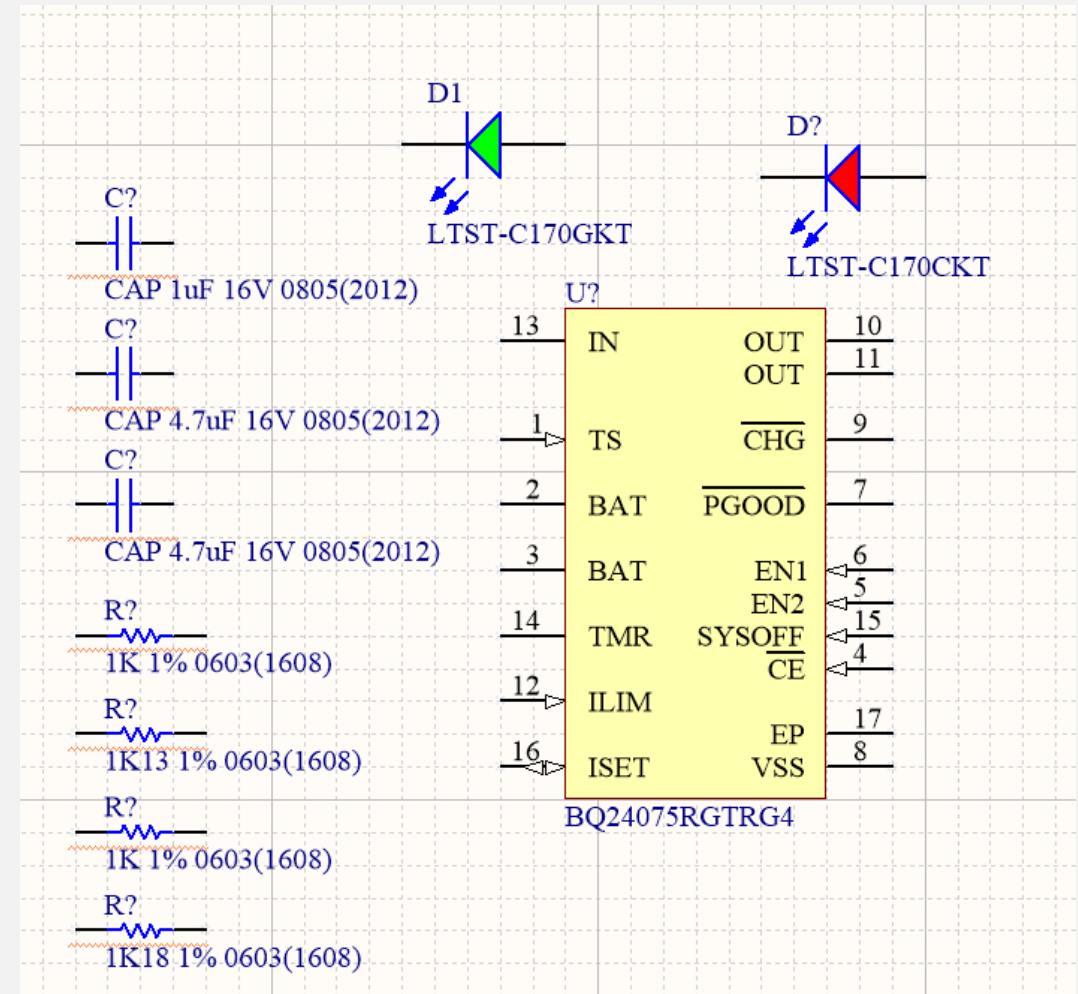
ADDING A COMPONENT FROM THE ALTIUM VAULT

- Great! Altium Content Vault has the part, saving us a lot of time. Right click the part and press “Place...”
- You should place this IC on the “LiPo_Charger” schematic you just did.
- Once you place the component, the Explorer window will be set back into focus again.



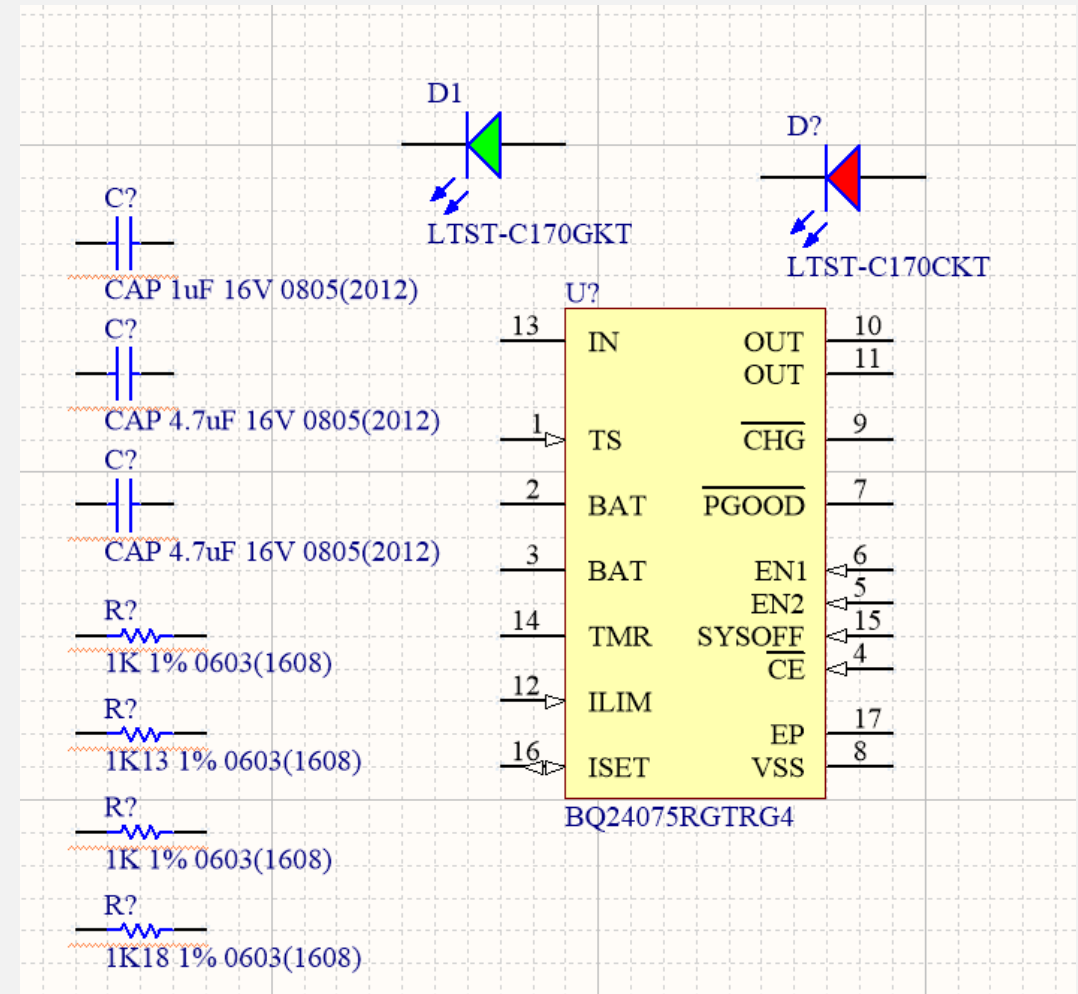
PLACING ALL THE OTHER COMPONENTS

- Now, we need to add the other components to make the suggested circuit. Please find the following components and place them on your sheet – do not worry about orientation or placement, we will worry about that later!
- LED1(for PGOOD): LTST-C170GKT
- LED2(for CHG): LTST-C170CKT
- 1uF Cap: Search for CMP-I036-04749-I
- 4.7uF Cap(x2) Search for CMP-I036-04893-I
- 1k resistor(x2)” Search for CMP-I012-00510-I
- 1.13k resistor: Search for CMP-I012-00515-I
- 1.18k resistor: Search for CMP-I012-00517-I



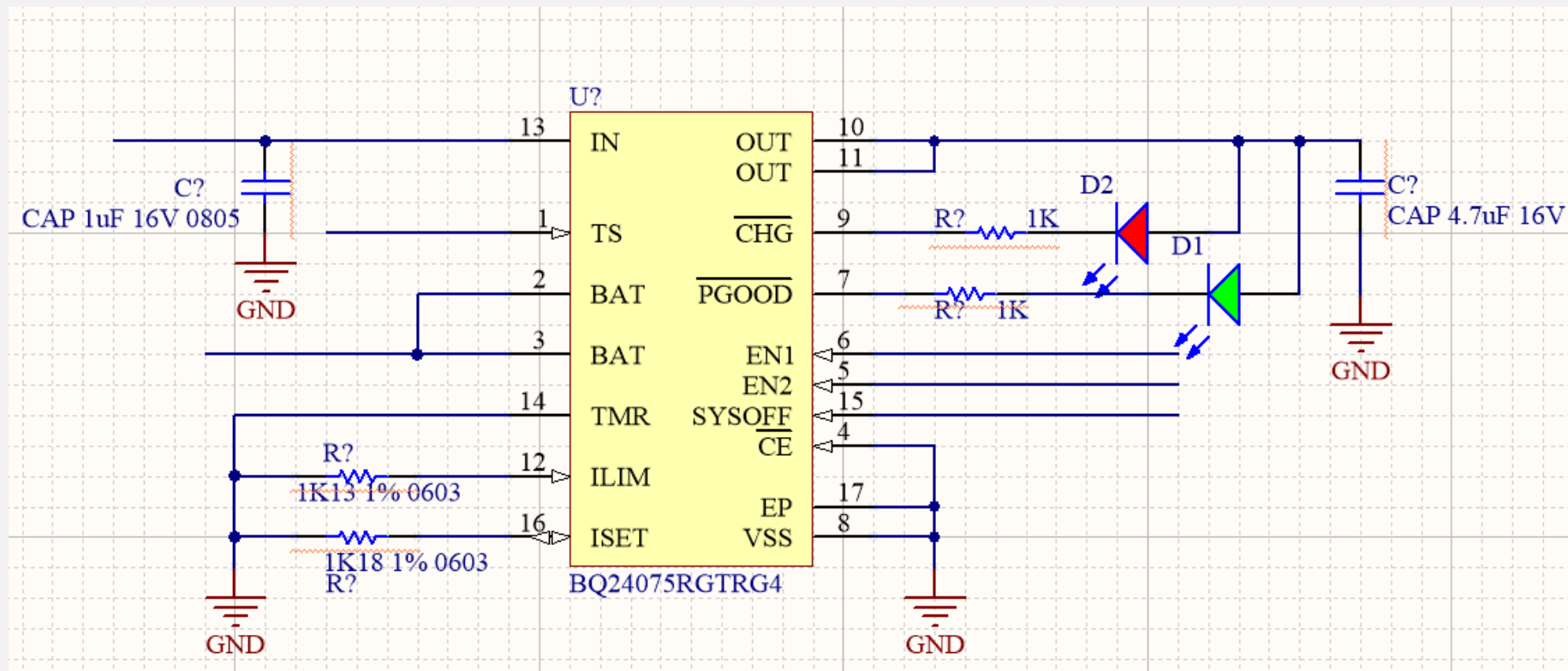
PLACING ALL THE OTHER COMPONENTS

- Now, we need to add the other components to make the suggested circuit. Please find the following components and place them on your sheet – do not worry about orientation or placement, we will worry about that later!
- LED1(for PGOOD): LTST-C170GKT
- LED2(for CHG): LTST-C170CKT
- 1uF Cap: Search for CMP-I036-04749-1
- 4.7uF Cap(x2) Search for CMP-I036-04893-1
- 1k resistor(x2)” Search for CMP-I012-00510-1
- 1.13k resistor: Search for CMP-I012-00515-1
- 1.18k resistor: Search for CMP-I012-00517-1



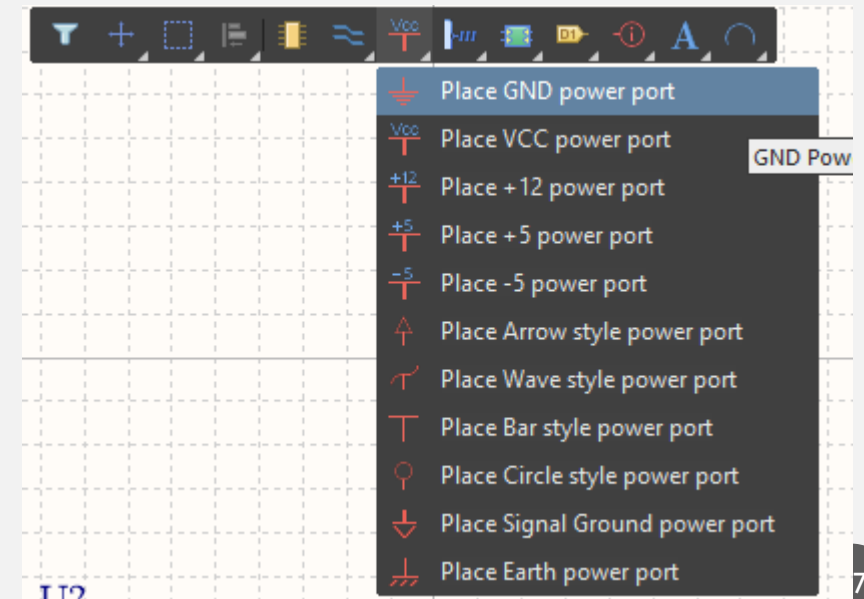
ORGANIZING AND WIRING COMPONENTS

- Now that you have the components, organize them and wire them similar to the picture. Tips on how to do this are on the following slide. Some wires appear to go nowhere – How we will use this will become apparent later on! **note: I forgot to add the 1uF capacitor on the BAT pins. Please do so yourself! The error is fixed in later slides.**

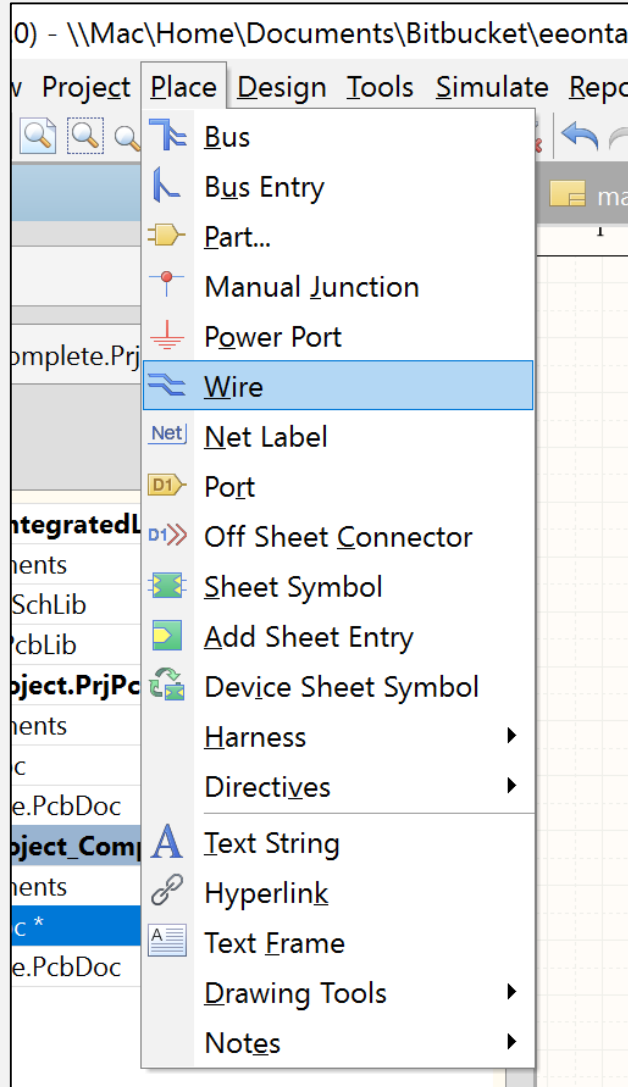


ORGANIZING AND WIRING COMPONENTS

- **Tip:** Grab components with the mouse and while dragging, press “control+spacebar” to rotate the component
- **Tip:** Use “Place->Wire” to wire components. However, it is much better to remember the keyboard shortcut since we do this a lot – “P” the “V” (how to remember: “P” of place, “V” of wire).
- **Tip:** To place GND, go to the toolbar and hold click on the port symbol (picture may be different) and choose Place GND power port.
- **Tip/Good practice:** Altium places a long comment string on all components with a lot of information that clutter the screen. It is a good practice to just leave the info that would be important for somebody checking the layout.
 - Example: For the resistors on the LED, the precision of the resistor and footprint is not important, so we erase that and only leave the value, which is important. For the capacitors, the value AND the voltage rating is important, so we leave those.
- **Tip/Good Practice:** Put text in a place that is readable! DO NOT leave text hidden in wires. In the context of this class it will be penalized, and it is a horrible practice to do so. Readability is key!



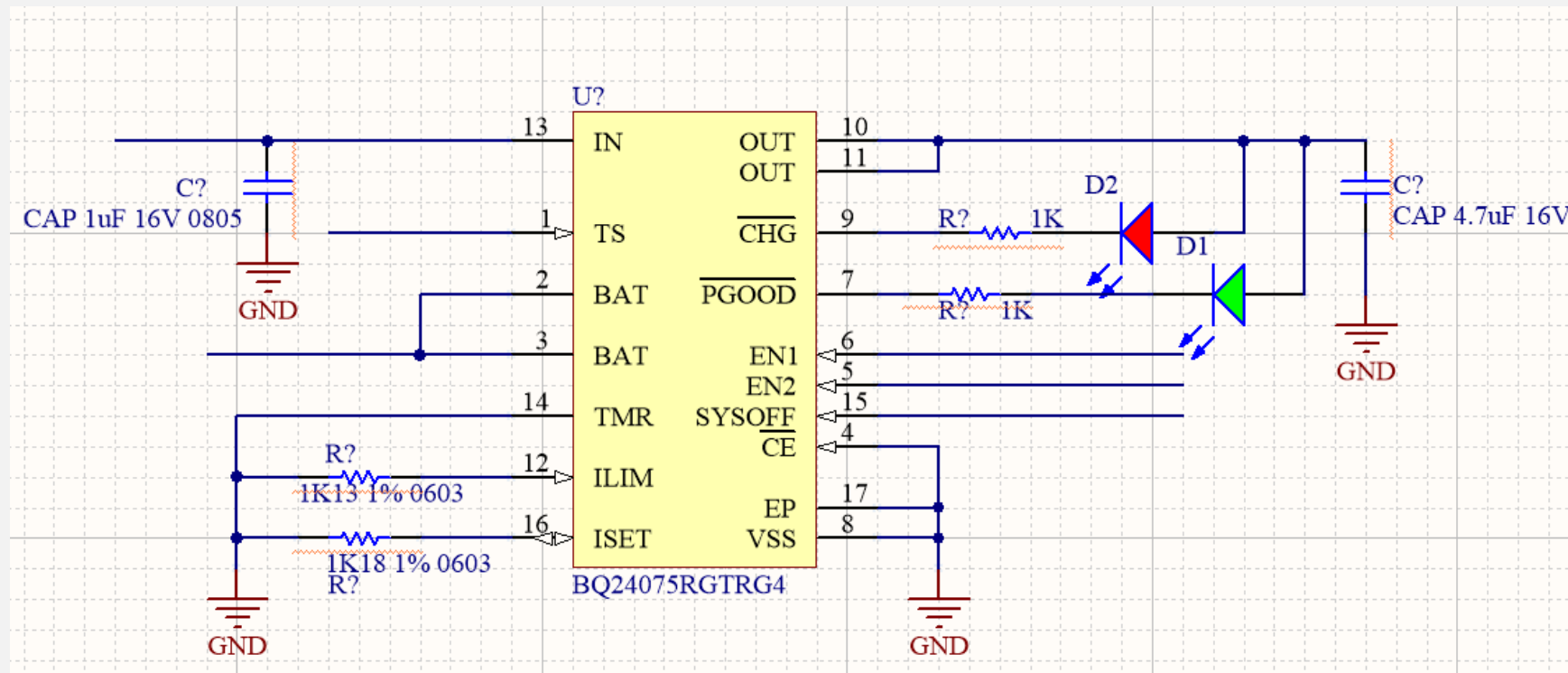
SCHEMATIC CAPTURE BASICS (FROM PREVIOUS YEARS)



- **Bus** – Route many wires together
- **Power port** – Add power or ground
- **Wire** – Connect two components
- **Net label** – Names a wire connection
- **Port** – Connect to a different sheet
- **Sheet symbol** – Used at the top level design to designate a lower level schematic sheet
- **Add Sheet Entry** – Entrance or exit to a lower level schematic sheet, corresponds to the **Ports** on the lower level
- For all of these commands, when active, you can press **Tab** to modify the options.
- Need help? Press **F1** while hovering on the button or panel you have a question about to bring up Altium's documentation.

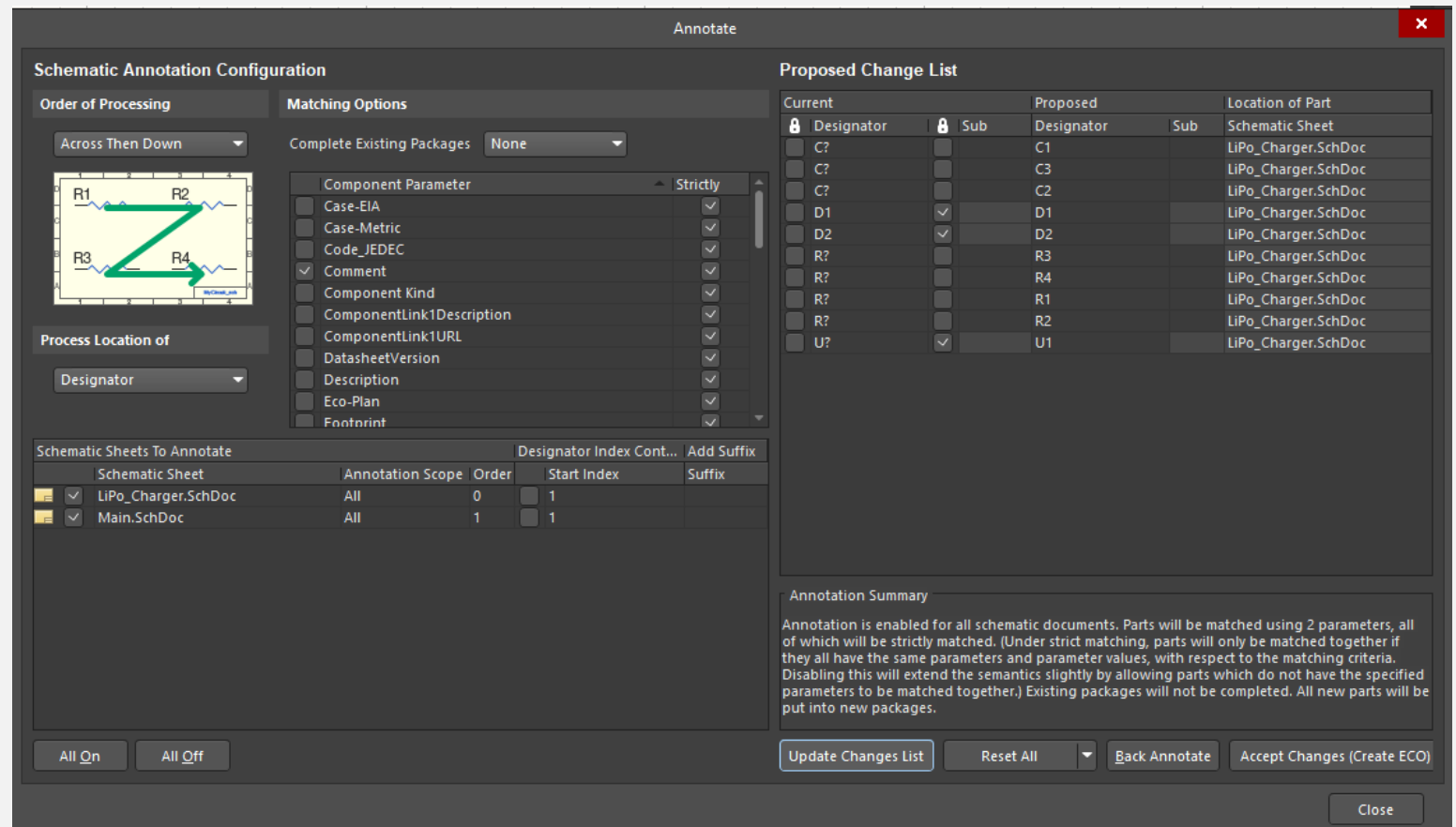
ORGANIZING AND WIRING COMPONENTS

- Now that you have the components wired, you might have noticed that the designators have an “?” attached to them, and a squiggly red line is beneath them. This is Altium telling us that there is an error – components must have a unique designator! You can change them manually, or you can do the process on the following slide.



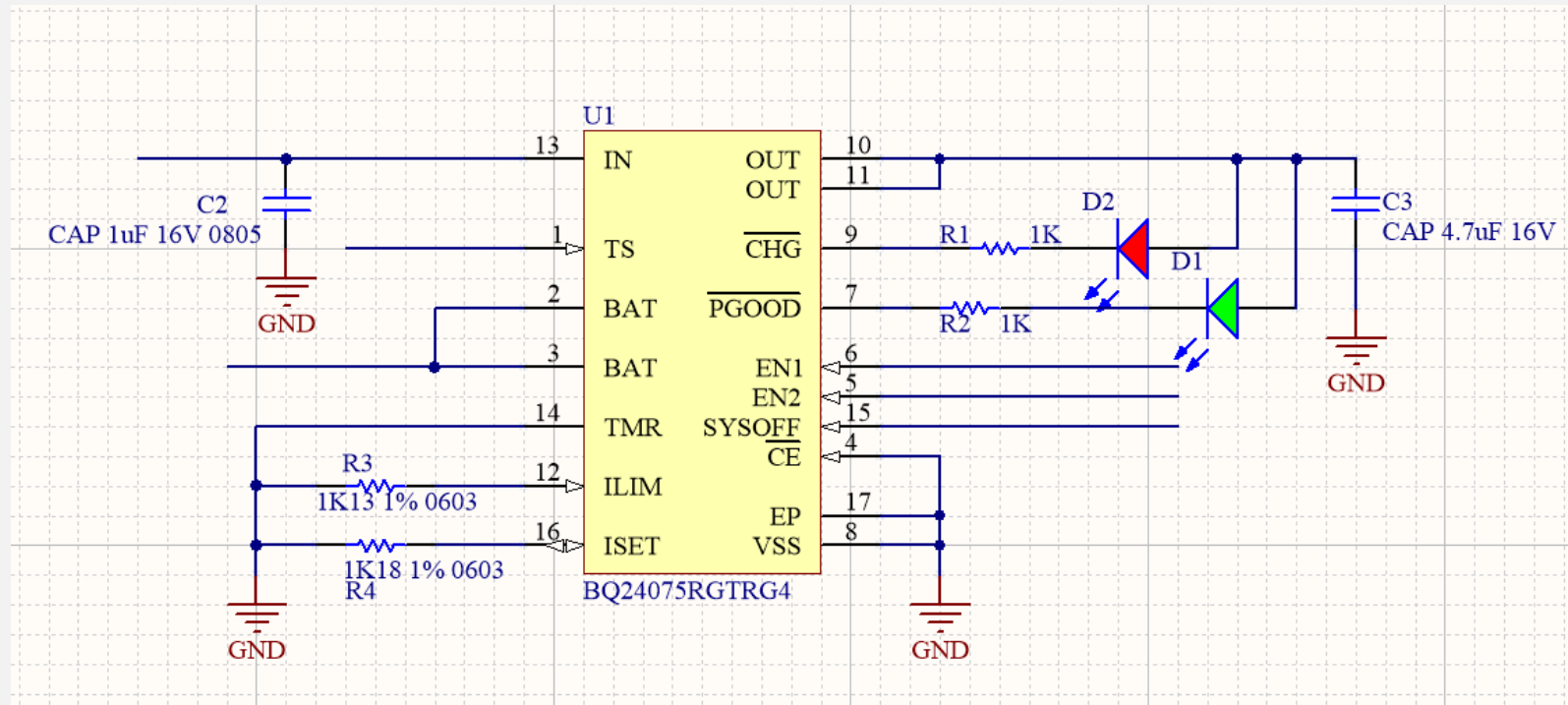
AUTOMATICALLY NAMING COMPONENTS

- Go to “Tools -> Annotation -> Annotate Schematics...”
- On the Annotate window, hit the “Update Change List” button. This will tell Altium to come up with names, as it will show you on the Proposed Change List. Hit “Accept Changes (Create ECO)”.



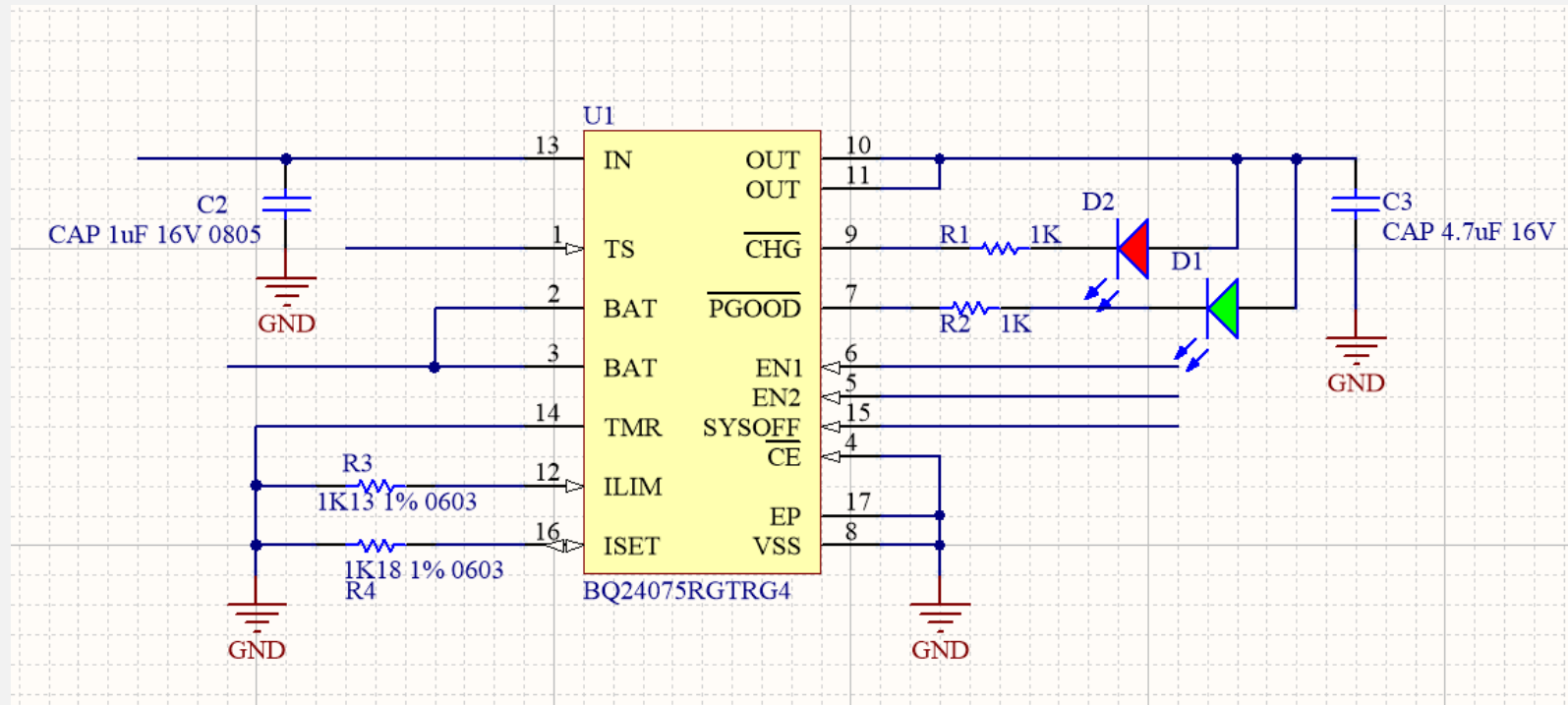
AUTOMATICALLY NAMING COMPONENTS

- Hit “Validate Changes” then “Execute Changes”. If everything went OK you should get green checkmarks. Hit Close. Your schematic should now have unique designator for all components (it might vary from this picture).



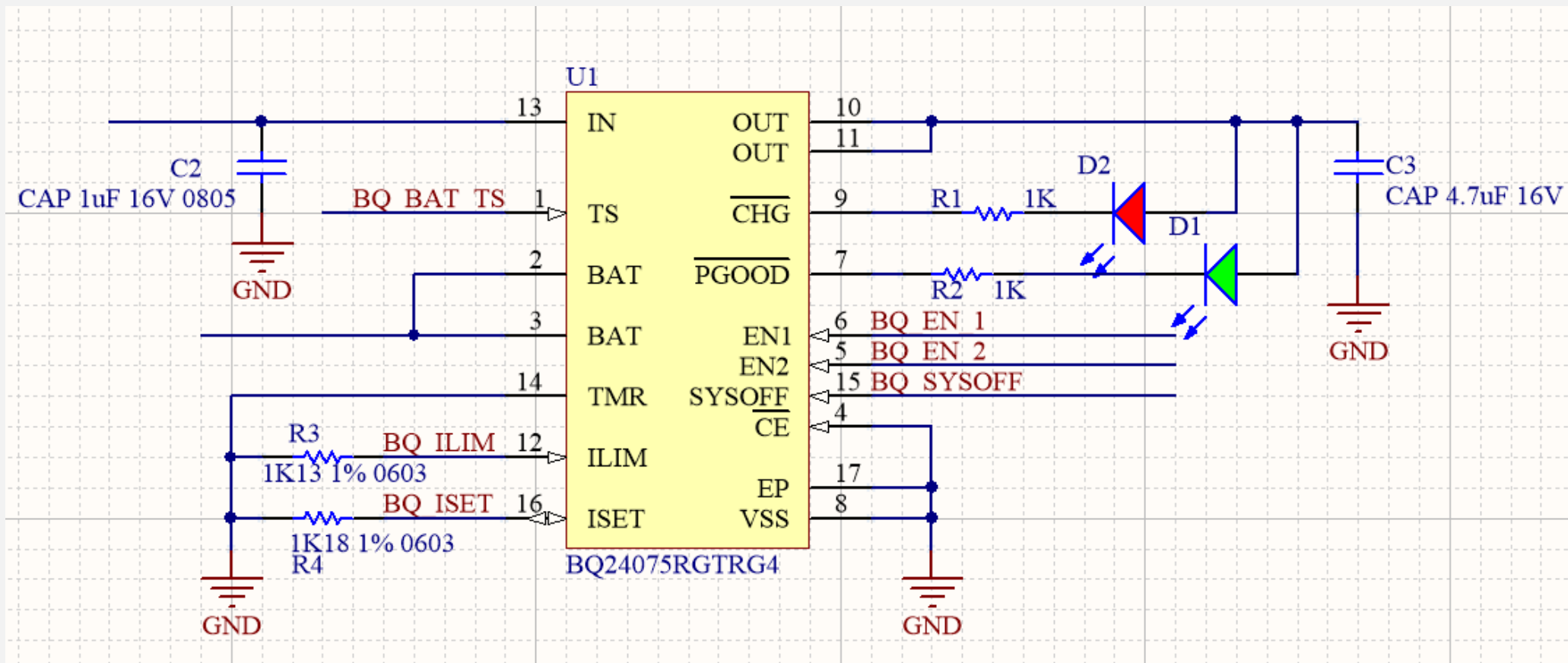
INTRODUCING NET NAMES

- Altium has a feature called Net Label (or Net Names) which greatly simplify how we make circuits.
- You can name a two wires with an identical Net Name and Altium will treat them as if they were connected (i.e. They are the same net).
- To place a net name, we use “Place->Net Label” or “P” followed by “N”.
- Good Practice:** We name most wires with a descriptive net name. Usually we use ALL CAPS separated by underscores. Power ports are an exception – we use another construct for them



INTRODUCING NET NAMES

- Please place net names on the wires as show on the example.

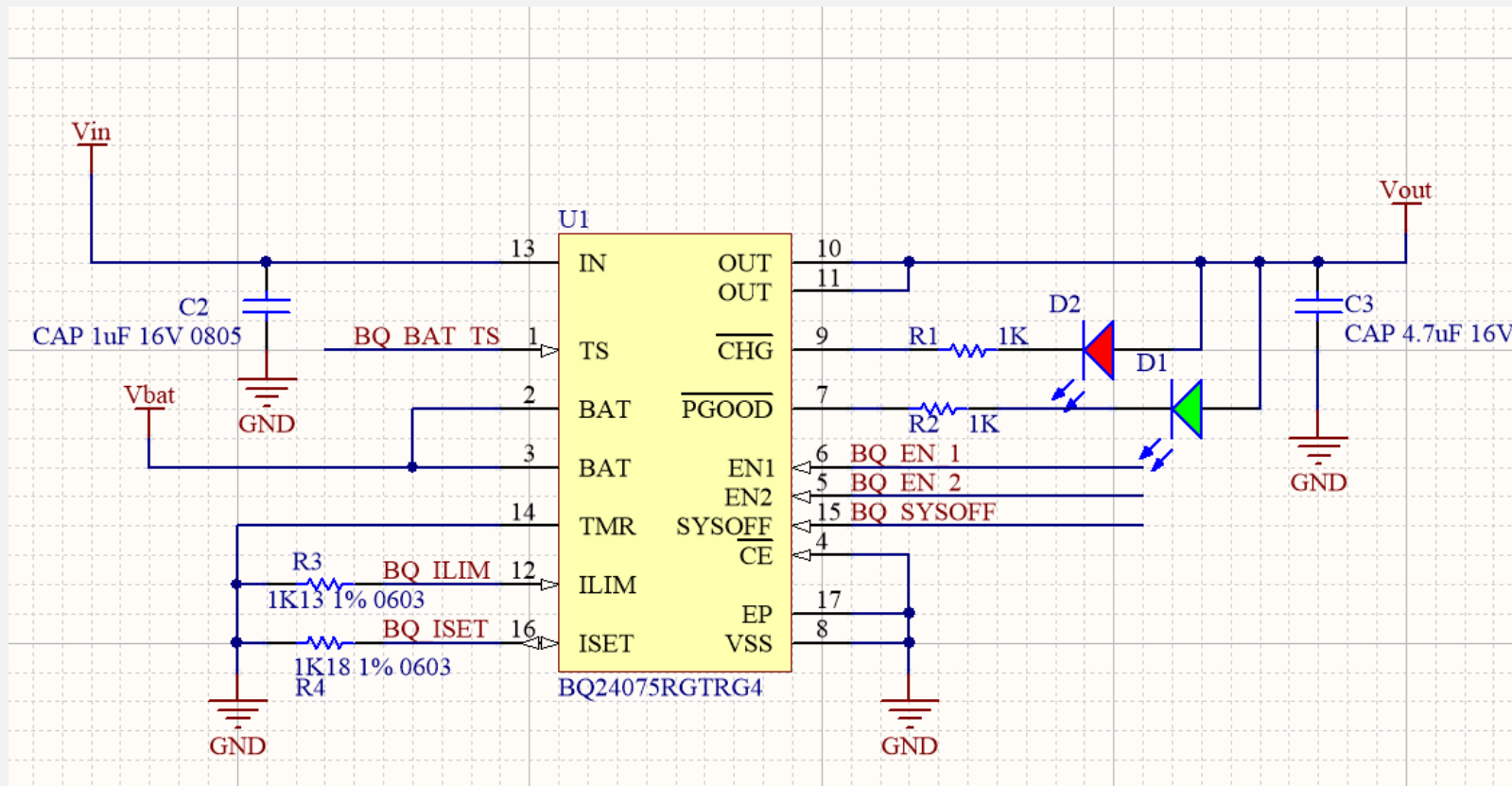


PLACING POWER PORTS

- For voltages, we have the construct of Power Ports. Power Ports are usually used globally throughout a project (net names can be “local” to the schematic that they use). This can be changed though in the project settings, but let’s leave it at the default use for now.
- We have three power ports used here:
 - VIN,
 - VBAT
 - Vout
- **Tip:** To place a power port, go to the toolbar and hold click on the port symbol (picture may be different) and choose Place a VCC power port. You can change the name to an unique name to differentiate between VCC ports.

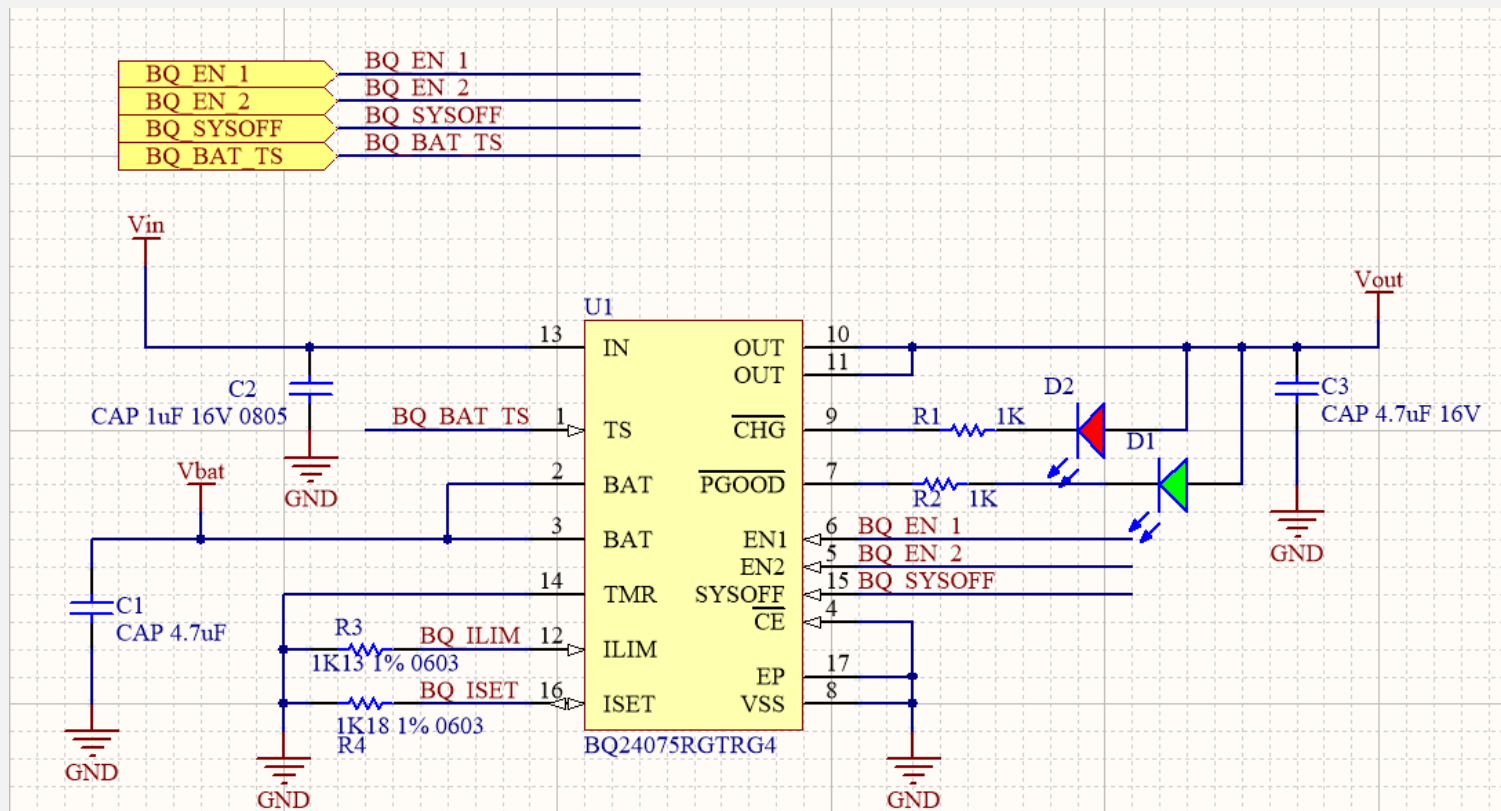
PLACING POWER PORTS

- Please power ports as show on the example.



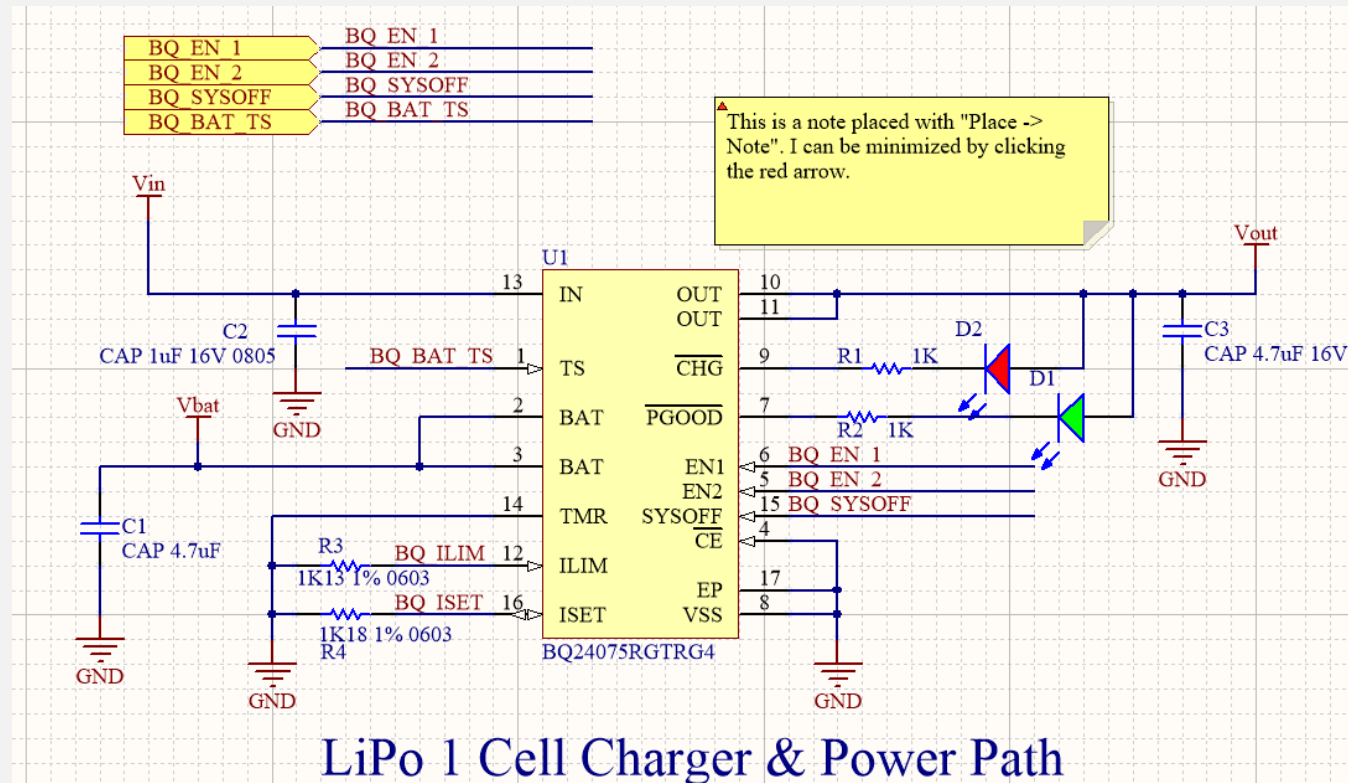
PLACING POWER PORTS

- Please place four ports: One for EN1, for EN2, for TS and another for SYSOFF. You can place a port with “Place -> Port”. Put wires on the ports and Add the same Net Names as the IC pins. This will connect the ports to the Ics without having to draw the wire between them, making the drawing way more clearer than if we did!



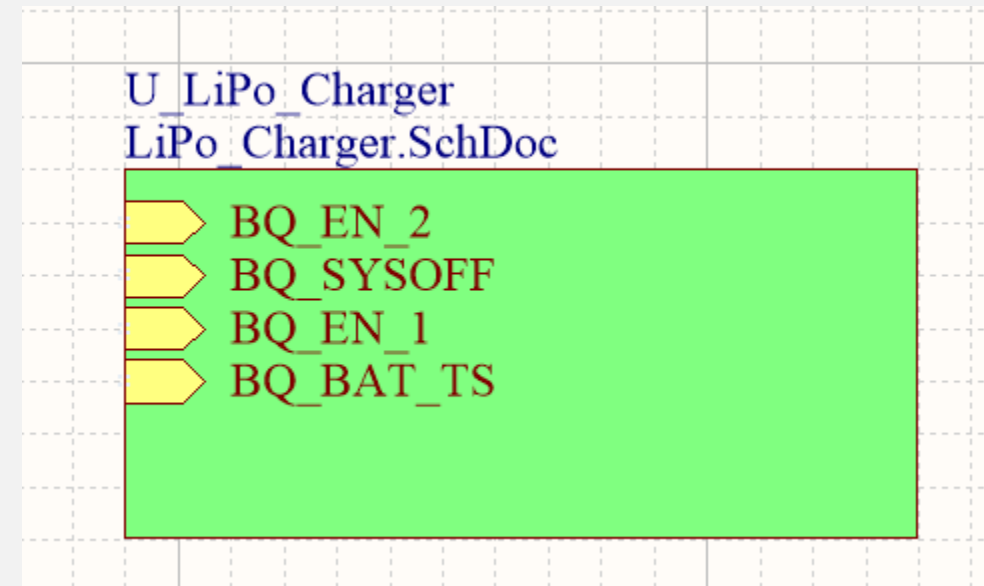
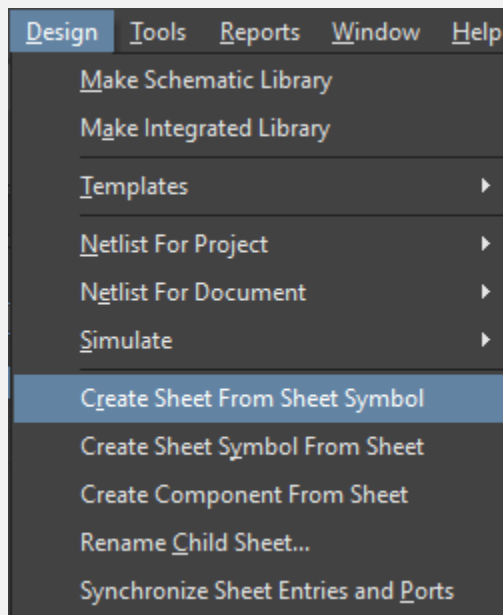
FINISHING TOUCHES

- It is good practice to put text with a title of the circuit you drew. You can also put notes/annotations with important information (Place – Text) (Place – Note). Save the schematic when you are done.



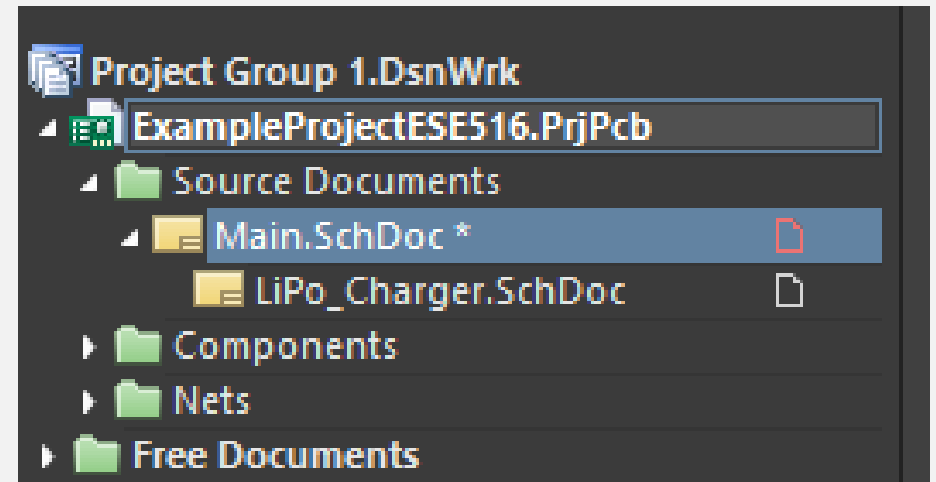
ADDING A SHEET SYMBOL TO ANOTHER SHEET

- We now have our sub-circuit, the LiPo charger, which we can add to another schematic file as a sub circuit. Altium calls this “Sheet Symbol”.
- Open “Main.shcdoc” on the Project Tab. Once open, go to “Design -> Create Sheet Symbol from Sheet”. Choose “LiPo Charger” and hit “Ok”. You can now put the sheet symbol of the sheet you just made!



SAVE YOUR PROJECT AND COMPILE

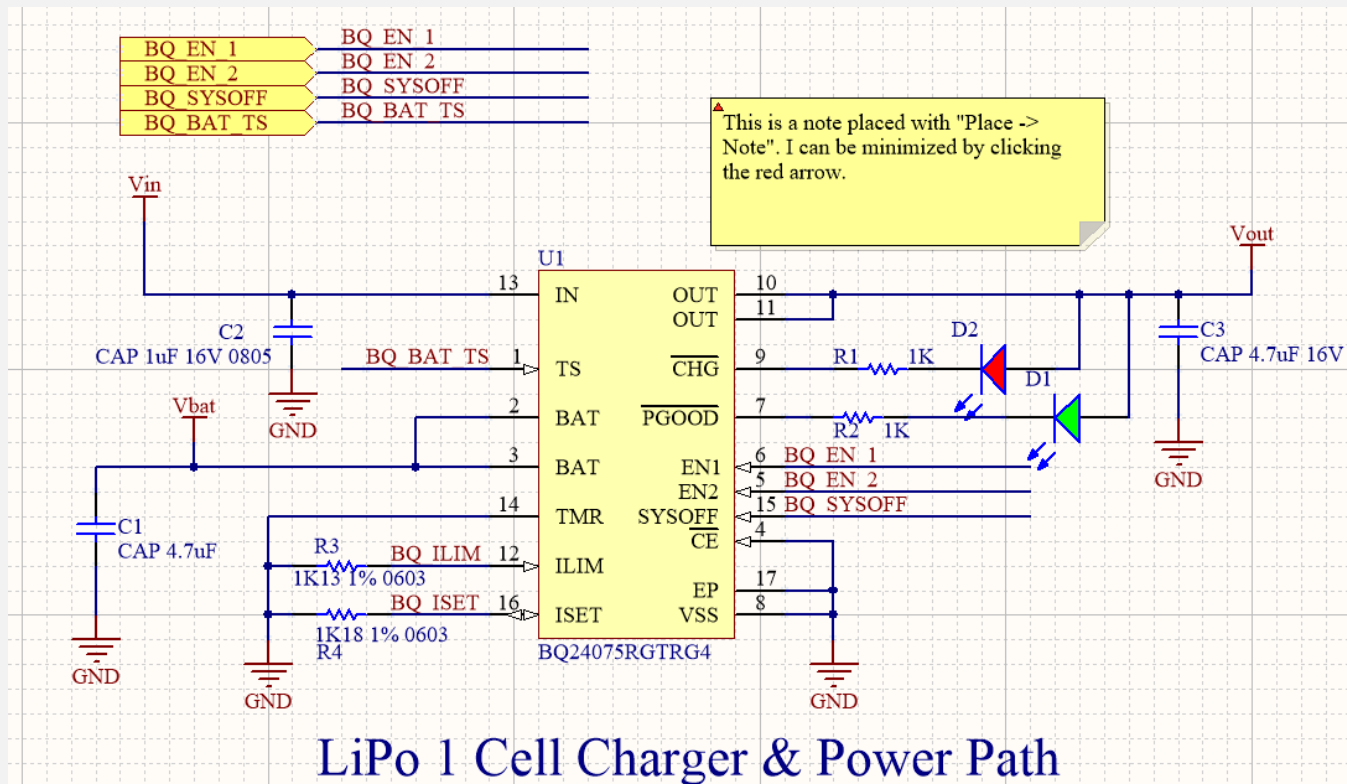
- Right click on your project and hit Save.
- To update the schematic to the new hierarchy, right click on the project and hit “compile Pcb Project <NAME>”. A new window with messages will appear – please ignore it.
- Altium will notice that you are using a schematic as a component for another schematic and will update the project tree. The schematic “LiPo_Charger” will now be under “Main”, showing the dependency.



SAVE YOUR PROJECT AND COMPILE

- Congratulations! You now know how to make a schematic of a sub circuit and use it for another schematic to make modular schematics!
- What about the power ports being global? Please read the following to better understand the different ways of setting them up: [https://www.altium.com/documentation/15.1/display/ADES/\(\(Creating+Connectivity\)\)_AD](https://www.altium.com/documentation/15.1/display/ADES/((Creating+Connectivity))_AD)
- **Please do read all the page! It solves a lot of details on how Altium can handle projects!**

CONGRATULATIONS



- Congratulations! You just have finished making a schematic (and reading the page I linked in the previous slide).
- It will get quicker with practice.